

2020

## **Development of FEM laboratory modules for an introductory undergraduate Geotechnical Engineering course**

Oswaldo P. M. Vitali

Amy Getchell

Marika Santagata

Follow this and additional works at: <https://docs.lib.purdue.edu/civeng-instres>

---

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries.  
Please contact [epubs@purdue.edu](mailto:epubs@purdue.edu) for additional information.

# **Development of FEM laboratory modules for an introductory undergraduate Geotechnical Engineering course**

Oswaldo P. M. Vitali<sup>1</sup>, Amy Getchell<sup>1</sup>, Marika Santagata<sup>1</sup>

<sup>1</sup>*Lyles School of Civil Engineering, Purdue University, West Lafayette, IN 47907, USA*

## **Abstract**

While geotechnical and structural engineers routinely rely heavily on finite element (FEM) modeling for analysis and design, exposure to numerical modeling and to software used in industry is generally limited in most civil engineering undergraduate curricula. This document summarizes the experience of introducing an industry FEM software - Midas GTS NX – into the Geotechnical Engineering I course at Purdue University in the Spring 2020 semester, through two laboratory modules built around key topics covered in the course: 2D groundwater flow and stress analysis. In addition to providing exposure to tools used in practice, the modules were designed to consolidate the comprehension of fundamental concepts, and enhance students' learning experience by facilitating rapid iterative exploration of different technical scenarios in the context of practical applications of geotechnical engineering.

## **1 Introduction**

Thanks to recent advances in software, hardware and constitutive modeling, geotechnical design currently relies heavily on numerical analyses, and most geotechnical and structural design firms have access to sophisticated finite element method (FEM) programs and the required computational capabilities, enabling highly complex numerical analyses to be conducted within tight project schedules. Numerical modeling for geo-structural design is highly attractive because it considers important aspects that may be neglected in traditional design methods, including coupled stress-seepage analyses, consideration of soil-structure interactions, realistic modeling of construction operations, and representation of complex aspects of soil behavior through advanced constitutive models, while taking into account complex 3D geometries and geological profiles.

With this in mind, there is great interest in introducing numerical modeling in the engineering curriculum at both the undergraduate and graduate level, and in particular in providing students with access to FEM software used in industry. These programs are generally user-friendly and have outstanding plotting features, permitting rapid visualization of results and exploration of realistic “practical” problems, which, a recent review of the state-of-the-art of geotechnical engineering education (Wirth et al. 2017) suggests, is critical even in an introductory geotechnical engineering course.

This report describes the laboratory modules developed using an industry-based FEM software (Midas GTS NX) for CE383: Geotechnical Engineering I, the introductory geotechnical engineering course offered at Purdue University during the Spring 2020 semester. This effort was motivated by a desire to increase student engagement; expose undergraduate students to tools used in practice; improve student learning experience by addressing problems more complicated than those typically tackled in the course, while also providing some exposure to design; and provide a more modern view of the geotechnical engineering field.

As part of this work, three lab modules were designed around key topics covered in the course, two of which are described in this report. The first addresses the 2D problem of seepage underneath a dam; the second the increase in stresses generated by the application of loads at the surface.

## **2 Background on course and student population**

Purdue’s CE 383 - Geotechnical Engineering I course is designed to introduce students to the fundamentals of soil mechanics with emphasis on the following topics: origin, composition and multi-phase nature of soils; effective stress principle; permeability and seepage; one dimensional consolidation theory and settlement analysis; strength behavior of coarse and fine grained soils. The course includes both a lecture (3 hours/week) and a laboratory component. Student assessment is based on weekly homework assignments and laboratory reports, in-class quizzes, exams and class participation. For the laboratory component, the students are divided into smaller sessions (generally < 24-26 students) that are held weekly for 2 hour periods and administered by the teaching assistant (TA). The laboratory sessions are intended to reemphasize the concepts

presented in the lectures, provide hands-on experience, and connect the class material to practical applications.

CE 383 – Geotechnical Engineering I is the first course in geotechnical engineering offered at Purdue University and, therefore, represents for all students the first academic exposure to this field. In Spring 2020, the class was comprised of 104 juniors and seniors and 1 graduate student, with 84 students enrolled in the Lyles School of Civil Engineering (CE), and 20 in the Division of Construction Engineering and Management (CEM). In the existing Purdue CE undergraduate curriculum, the course is not required for graduation (this is scheduled to change in Fall 2021). It is, instead, a required core course for the Purdue CEM students. In Spring 2020, none of the students enrolled in CE 383 – Geotechnical Engineering I had any previous experience with numerical methods, including FEM.

### **3 FEM Laboratory module design and implementation**

#### **3.1 Overview**

While focusing on different topics, the two laboratory modules described in this report were both designed and administered in a similar manner. The FEM software Midas GTS NX was made available to all students to download to their personal laptops to use during the FEM laboratory sessions. Additional laptops on which the software had already been installed were provided during the laboratory session to those who did not have a laptop or could not successfully install the software. In both labs, students worked in pairs or, in exceptional cases, in groups of three. This was done to encourage peer to peer interaction and team work. The assignments were designed to be completed during the 2 hour lab session with assistance from the TAs. Student assessment was based on responses to a series of questions included in a worksheet distributed to the students and on engagement during the lab session.

For each FEM laboratory module, the following educational materials were developed by the teaching team and provided to the students:

- (1) a “master” handout, with detailed explanation of the problem and a step-by-step procedure to guide students through the lab;

- (2) one or more FEM models (Midas GTS NX input files) with mesh, boundary conditions and material properties set up;
- (3) a short tutorial (software specific) illustrating how to conduct the analysis, visualize and extract the results, and, when needed, modify the model;
- (4) a worksheet, with questions designed to focus students' attention on the most important aspects of the problem;
- (5) an Excel spreadsheet with graphs ready for plotting select data extracted from the FEM output.

Additionally, a grading rubric was developed for use by the teaching team.

In both labs, the approach was to first analyze a problem using traditional methods covered in the lectures. Then, the same problem was analyzed using the FEM software, and the students were asked to compare the numerical results with their calculations. After this initial task, the students used the FEM software to explore more complex scenarios that could not be readily solved using hand-calculations or available solutions. The FEM models were set-up ahead of time, and provided to the students, who were instructed to inspect them, and review the FEM mesh, the boundary conditions including load/head distributions and the material properties. This was done so that, given the limited time available during the lab session, the students could focus on the fundamental concepts instead of on modeling procedures and implementation. The master handout guided the students through the assignment, and directed them to answer the questions on the worksheet as they progressed. The questions were designed to consolidate the fundamental concepts that had been introduced in lecture and focus students' attention on aspects that may not be obvious at a first glance. In a few cases the students were asked to predict the outcome of an analysis prior to performing it, and then encouraged to reconcile their intuition with the numerical results. In the last part of the lab the students were asked to build on the work performed to that point to address a design question. This was done to emphasize the role of the FEM as a practical tool for engineering design. The course did not allow the time to discuss the theory of the finite element method. However, ahead of the second FEM lab a lecture was offered to introduce the philosophy of the method (e.g. concept of discretization, basic definitions, assembly of stiffness matrix), and share some examples of its application in practice in the field of civil engineering.

The following two sections provide more detail on the two FEM laboratory modules implemented during the 2020 Spring semester:

- (1) Laboratory on 2D seepage using FEM (administered in the course as Lab 5)
- (2) Laboratory on stress analysis using FEM (administered in the course as Lab 8)

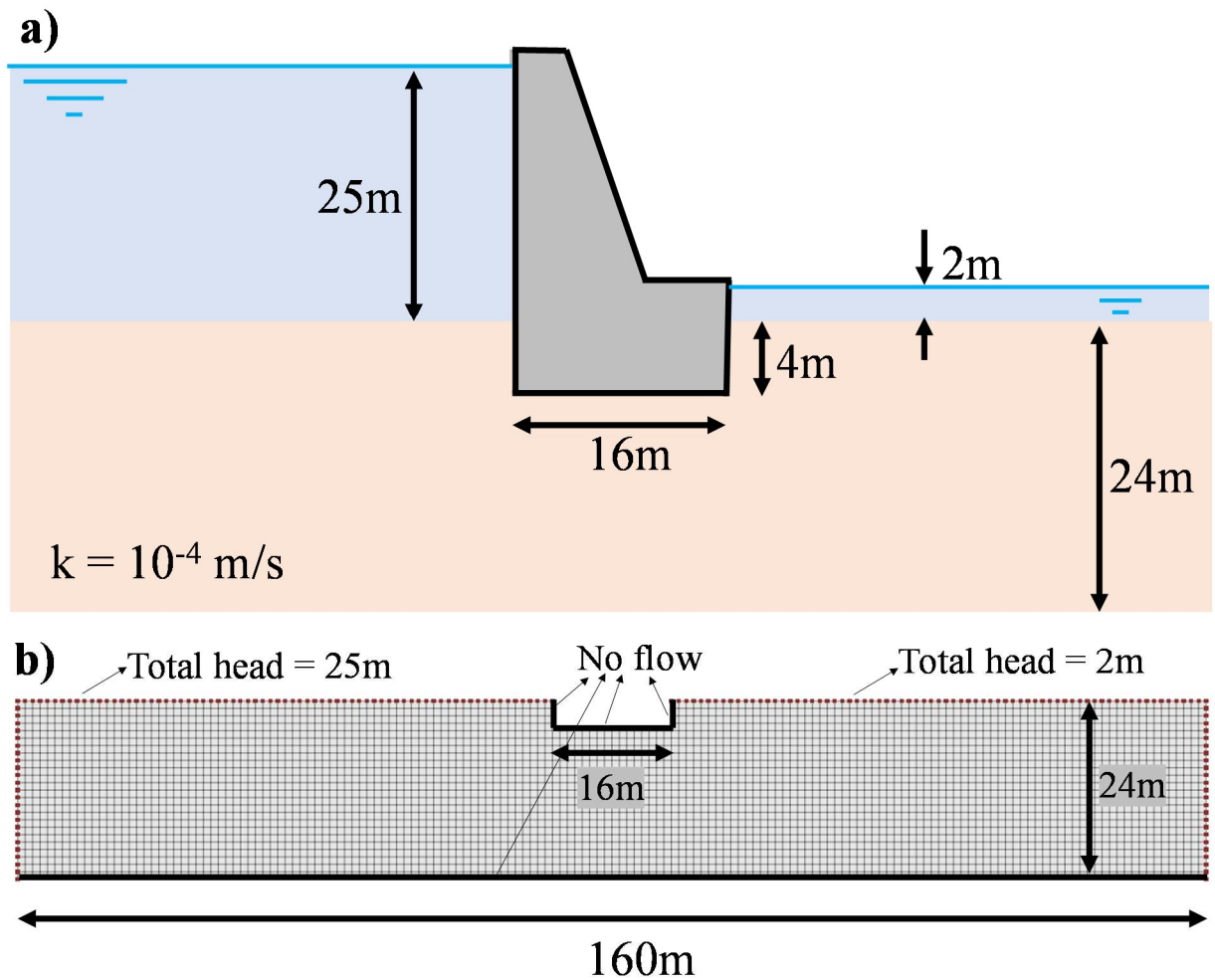
The materials developed for these two modules are attached as Appendices.

### **3.2 Laboratory on 2D seepage using FEM**

The FEM laboratory module on 2D groundwater flow explores the practical problem of seepage underneath a dam. The module was designed with the following key student learning objectives:

- become aware of the potential to solve seepage problems using the FEM;
- acquire familiarity with the terminology and functionality of a commercial FEM software;
- develop the ability to calculate fundamental quantities (e.g. gradients, flow rates) in more complex flow conditions than those previously explored in homework assignments;
- understand the relationship between the geometry of the flow paths and key results (e.g. flow rate, maximum exit gradient) produced by the analysis;
- examine the effectiveness of techniques used for controlling flow.

A sketch of the dam, with the geometry and water levels, and the corresponding FEM model are shown in Figure 1a and Figure 1b, respectively. The boundary conditions are applied at the ground surface at the left of the dam (upstream, total head of 25m) and at the right (downstream, total head of 2m). The ground surface is selected as the datum. No flow is allowed through the contour of the dam and through the bottom of the model. The ground is discretized using square elements, with sides of 1m. The width of the model (160m) was chosen based on preliminary analyses that showed that beyond this size the results were not significantly affected by the width of the model.



**Figure 1 Practical problem of groundwater flow underneath a dam. (a) Geometry with dimensions and water levels and, (b) finite element mesh.**

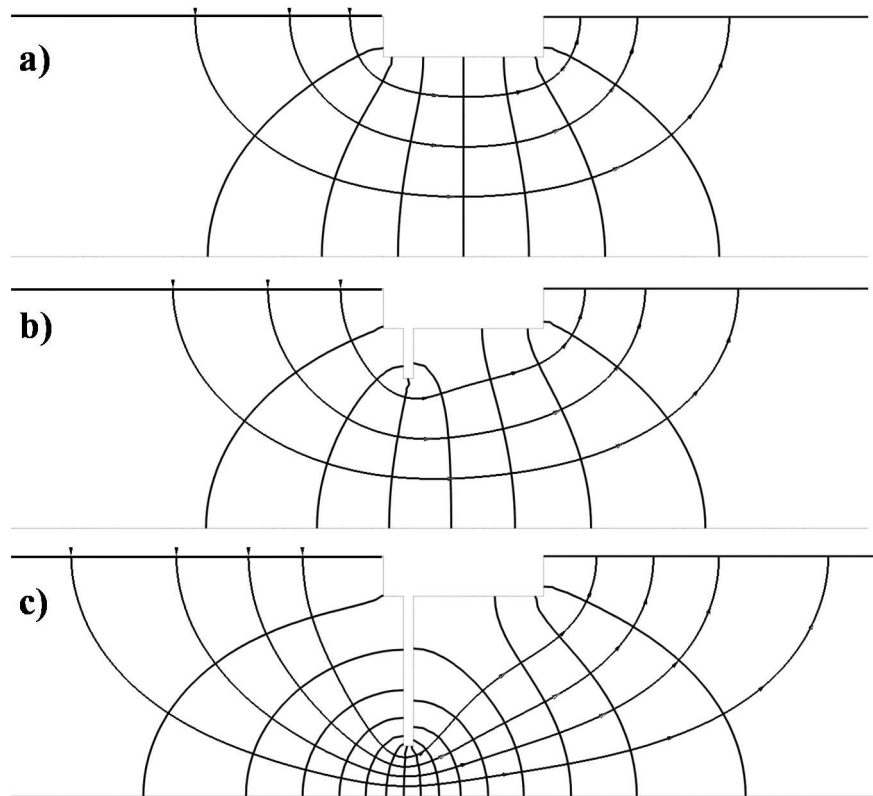
The laboratory module was organized in three parts:

In Part A, the students were asked to determine the maximum exit hydraulic gradient, the total flow rate, and the uplift force under the dam by hand using an already prepared flow net.

In Part B, the students first explored the same problem using the FEM software. The same quantities were extracted from the FEM model and compared with the values obtained with the flow net, so that the students could associate the numerical model with the flow net analysis. The

students were then asked to use the FEM software to analyze the effects of placing a grout screen of given depth under the upstream and the downstream end of the dam.

Finally, in Part C of the lab, the students were asked to use the FEM software to select the grout screen design (position and depth) that met specific design requirements (limit on maximum exit gradient, and minimum factor of safety against uplift).<sup>1</sup>



**Figure 2 Plots of the equipotential lines together with flow path lines visualized in Midas GTS NX: (a) dam with no grout screen; (b) dam with 5 m deep grout screen, and (c) dam with 15 m deep grout screen.**

During the laboratory session, the students were encouraged to visualize total head and pressure head fields and flow paths. They were also asked to build flow nets for the different cases examined by plotting together the equipotential lines and the flow path lines. Typical plots prepared by the students during the laboratory session following the instructions provided in the software tutorial

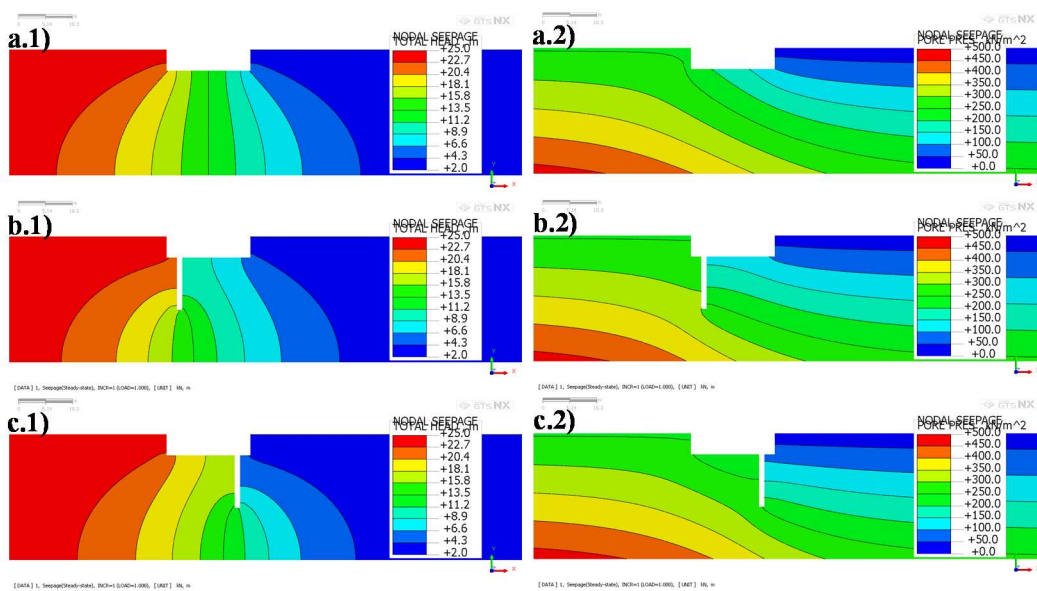
---

<sup>1</sup> This design question was further explored by the students in the subsequent laboratory session in which they utilized the finite difference method to analyze the same 2D seepage problem.



included in their packet are shown in Figures 2 and 3. Using the software, the students could quickly assess a variety of different scenarios, different grout screen depths and screen position to evaluate how the grout screen affected the flow paths and the total and pressure heads. Answering the questions included in their worksheet required the students to explore and understand these plots.

The materials developed for this module are included in Appendix A.



**Figure 3 Plots of the total head field (1) and pore pressure field (2) visualized in Midas GTS NX: (a) dam with no grout screen, (b) dam with 10 m deep grout screen near the upstream edge and, (c) dam with 10 m deep grout screen near the downstream edge.**

### 3.3 Laboratory on stress analysis using FEM

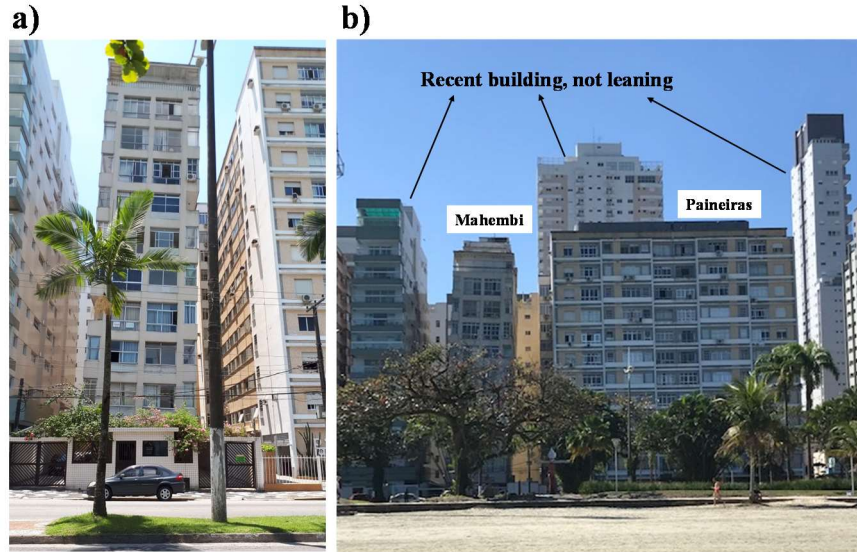
The FEM laboratory module on stress analysis addresses the change in stresses produced in a soil mass due to the application of loads at the ground surface. The learning objectives of this module were for students to:

- become aware of the potential to model stress analysis problems using the FEM;
- acquire familiarity with the terminology and functionality of a commercial FEM software;

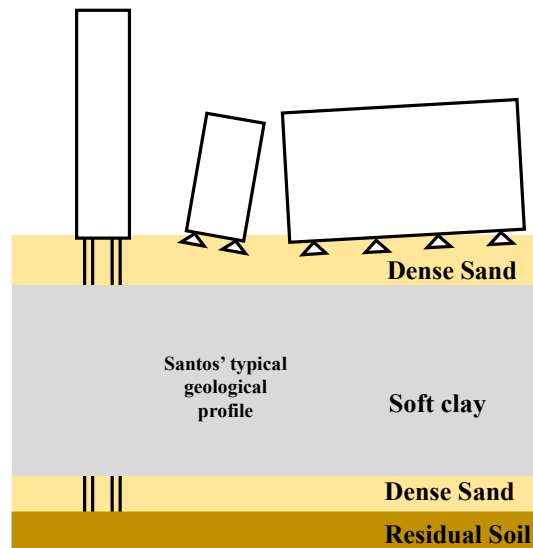
- reach an increased understanding of fundamental concepts including: superposition of elastic solutions; relationship between stress changes and settlement of a soil mass; stress propagation in layered soils;
- gain insight into the impact of foundation type on the performance of a structure.

The case history of the leaning buildings in Santos, SP, Brazil, was selected for this FEM laboratory module. Figure 4 shows pictures of the two leaning buildings, Mahembi and Paineras (Figure 4a), used in the laboratory assignment and three other more recent buildings that show no lean (Figure 4b). Students were provided with a typical simplified geological profile (Figure 5), which includes a superficial layer of dense sand (~ 10 m), a very thick (up to 50 m) layer of soft clay, a layer of dense sand, underlain by residual soil. They were also informed that the leaning buildings had been constructed on shallow foundations, while the level buildings were built on deep foundations (Figure 5).

The Mahembi and Paineras buildings are representative of a number of tall buildings constructed in Santos between the 1940's and the 1980's using shallow foundations, prior to the availability of practical technology to build deep foundations in Santos' geological profile. Concerns regarding the settlement of the buildings were raised by the early 1950's (Teixeira 1994) by geotechnical engineers, who warned that large settlements and leaning of the buildings would occur with time due to consolidation of the thick, soft clay layer underlying the superficial sand layer. A number of recommendations were provided, including limiting the number of floors of new buildings to ten (Teixeira, 1994). This recommendation was repeatedly disregarded, and buildings with up to 18 floors were built. It wasn't until 1986 that the Building Code of the Municipality of Santos stated that "Buildings with more than twelve floors must have deep foundations." Buildings with more than 30 floors have since been successfully constructed in Santos over deep foundations. This brief and simplified history of Santos' leaning buildings was shared with the students during a lecture offered ahead of the lab session to provide context for the laboratory exercise and emphasize the importance of geotechnical considerations in design. This case history also provided a link to the subsequent topic covered in the course: consolidation theory and settlement analyses.



**Figure 4 Leaning buildings in Santos, São Paulo, Brazil: (a) Mahembi building; (b) Mahembi and Paineiras buildings alongside more recent buildings showing no lean.**



**Figure 5 Typical geological profile in Santos with illustration of lean observed in buildings on shallow versus those on deep foundations.**

Figure 6 shows the FEM model used for laboratory exercise. For simplicity the problem was considered two-dimensional, i.e. modeled as a plane strain problem. The buildings were represented by a distributed load applied on the ground surface. The model dimensions were very large to avoid boundary effects and to allow a better visualization of the stress bulbs. The

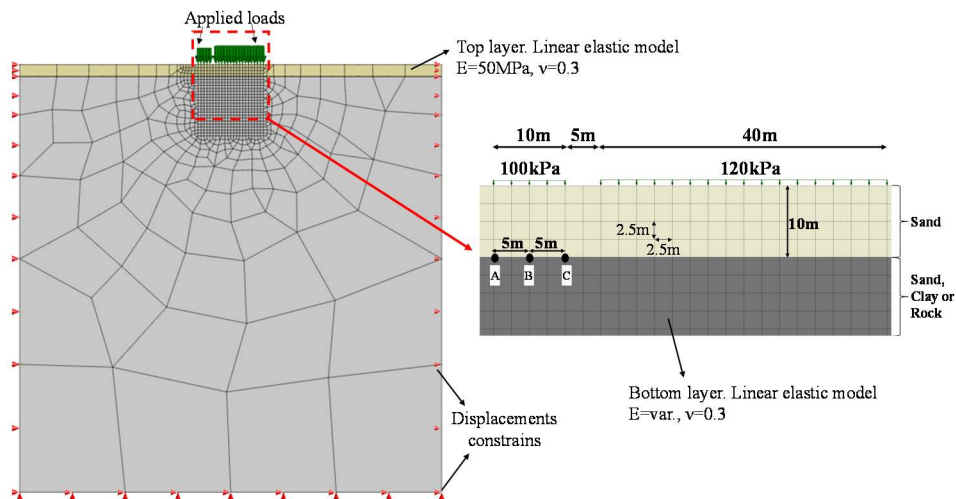
dimensions of the buildings were roughly determined from photographs and the loads estimated based on the number of stories. The ground was simulated with linear elasticity. Second order rectangular elements were adopted to ensure the accuracy of the numerical results. The in-situ stresses were neglected since the laboratory exercise focused on vertical stress increments in the ground. Two loading conditions were analyzed:

- *Loading condition 1* - “small building only,” which considers only the load of the small building (Mahembi);
- *Loading condition 2* - “small and large buildings,” in which the loads due to both buildings (Mahembi and Paineras) are considered.

Students were asked to consider the following three soil profiles which differ in the stiffness of the soil layer below a depth of 10m:

- *Soil profile 1* – uniform sand layer ( $E_{\text{sand}}=50\text{MPa}$  and  $\nu=0.3$ );
- *Soil profile 2* – 10m of sand underlain by soft clay ( $E_{\text{clay}}=2.5\text{MPa}$  and  $\nu=0.3$ );
- *Soil profile 3* – 10m of sand underlain by rock ( $E_{\text{rock}}=5\text{GPa}$  and  $\nu=0.3$ ).

*Soil profile 2* is intended to be representative of the actual geological profile in Santos.



**Figure 6 Numerical model used for study of problem of the leaning buildings in Santos.**

This FEM laboratory module was divided in five parts:

- *Part A*: Comparison between numerical results and closed-form elastic solution (based on *Soil Profile 1* - uniform sand layer, for both loading conditions)
- *Part B*: Effect of the large building on vertical stress increments produced under the small building (comparison between *Loading condition 1* - “small building only” and *Loading condition 2* - “small and large buildings,” considering *Soil Profile 1* – “uniform sand layer”).
- *Part C*: Effect of the presence of soft clay under the sand layer (comparison of *Soil Profiles 1 and 2* for both loading conditions).
- *Part D*: Effect of the presence of rock under the sand layer (comparison of *Soil Profiles 1, 2 and 3* for *Loading condition 1*).
- *Part E*: Design implications: estimate of maximum allowable number of stories to limit settlement under small building (based on *Loading condition 2* and *Soil Profile 2*).

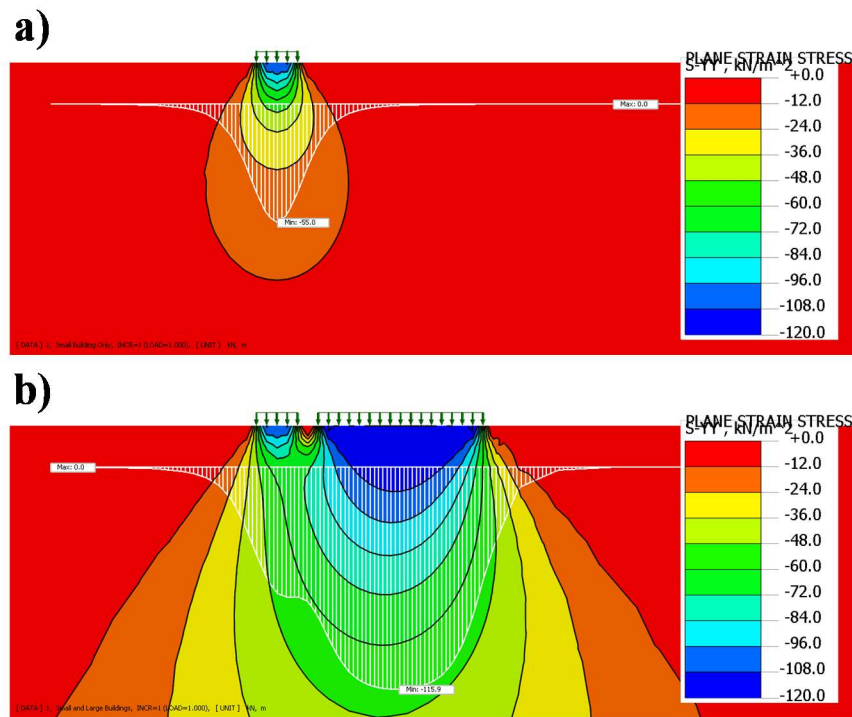
In Part A, the vertical stress increments obtained with the FEM model at points A, B and C (Figure 6) were compared to those calculated using the Boussinesq elastic solution during a previous laboratory session on elastic solutions.

In Part B, the students were encouraged to visualize and inspect the vertical stress increment bulbs for the two loading conditions examined, to associate the changes in the stresses underneath the small building with the observed lean. Figure 7 shows typical plots visualized by the students in this part. Students were also instructed to extract results of stress increments produced with the two loading scenarios at different points below the centerline of the small building, and at a depth of 10 m underneath the small building. They used these data to prepare plots in excel and answer questions related to the impact of the load from the large building. As in Part A, stress increments at the same reference points (A, B and C in Figure 6) were summarized by students in a table.

In Parts C and D, students assessed the effect of changing the material properties of the bottom (>10 m depth) layer on the vertical stress increments produced by the applied loads. Again, students were asked to visualize the stress bulbs and compare them to those observed for a uniform deposit (*Soil profile 1*).

Finally, in the last part of the lab (Part E), students explored the design implications of what they had learned. Focusing on *Soil Profile 2*, which is representative of Santos' geological profile, students were asked to provide an estimate of the maximum number of stories that they would have recommended for the larger (Paineiras) building to limit deformations under the existing small (Mahembi) building. As students were not yet familiar with settlement analysis in soft clays (this topic is taught later in the semester), the problem was simplified greatly and students were told that to limit large deformations, the soft clay could support a maximum vertical stress increment of 50 kPa. This number was selected based on the recommendation that emerged from the debates promoted by the *Associação Brasileira de Mecânica dos Solos – Núcleo Regional de São Paulo* in 1995, to limit the vertical stress increment at the top of the soft clay layer to less than 50 kPa (Massad 2009).

The materials developed for this module are included in Appendix B.



**Figure 7 Vertical stress increment plots: (a) Loading condition 1 - small building only and (b) Loading condition 2 - small and large buildings (both for *Soil Profile 1* – uniform soil).**

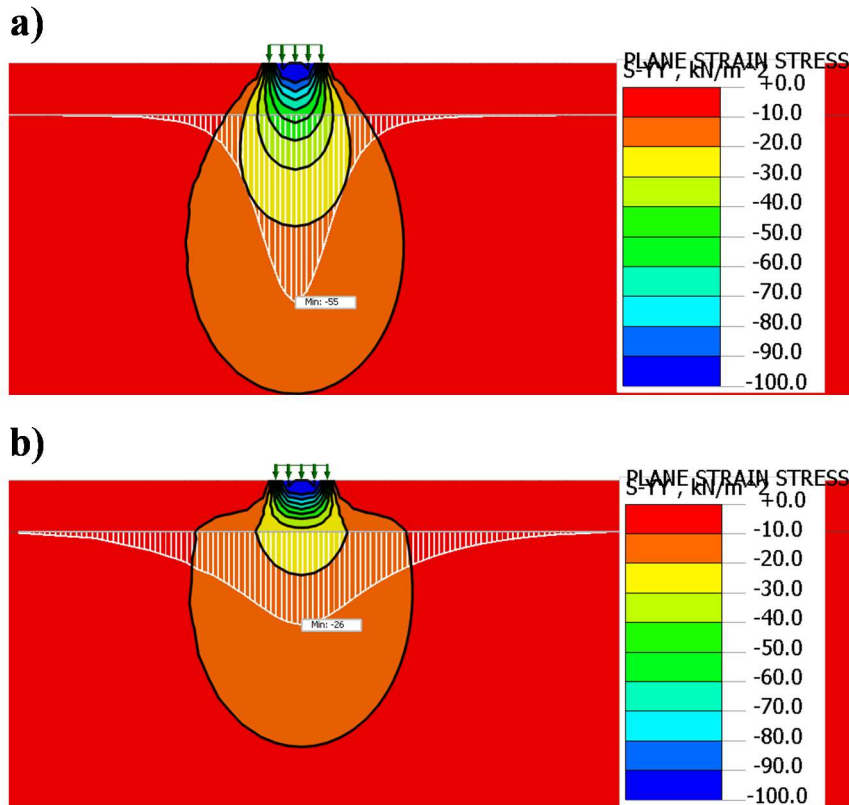


Figure 8 Vertical stress increment plots for *Loading condition 1* – “small building only” for (a) *Soil Profile 1* – uniform sand layer; and (b) *Soil Profile 2* –soft clay under sand layer.

## 4 Final remarks

This document summarizes the educational modules developed using an industry-based FEM software (Midas GTX NX) for the introductory geotechnical engineering course offered at Purdue University in the Spring 2020 semester. The two modules were administered as part of the laboratory component of the course. The modules, developed around two core topics covered in the course - 2D seepage and stress analysis – are connected to practical applications of geotechnical engineering: the design of a grout screen under a dam, and the effect of new construction on differential settlement under an existing building on shallow foundations.

In both exercises the students first analyzed a simple base case problem using both the FEM and traditional methods previously covered in class; then they used the software to explore more complex scenarios that could not be readily solved using hand-calculations or available solutions;

finally, using the understanding of the problem developed through this work they addressed a design question. The educational materials developed for these modules were designed to guide students through the assignment, and focus their attention on key fundamental concepts.

The access to a user-friendly FEM software with great visualization capabilities facilitated iterative exploration of technical parameters, enabling a depth of analysis that is not common in an undergraduate course, and providing students with a sophisticated understanding of the problems investigated in a short period of time.

While designed to be administered during a single two hour laboratory session, both laboratory modules described in this document could be split over two or more sessions and/or expanded to allow exploration of additional factors (e.g. the role played by material anisotropy) or to include a more comprehensive design exercise. In addition, the second of the two modules could be revisited after covering the topic of consolidation for calculation of the vertical strain field and surface settlement.

## **Acknowledgments**

The authors acknowledge the generous support from the MIDASoft company, which provided academic licenses of the FEM software Midas GTX NX used in the FEM laboratory modules.

## **References cited**

Massad FM (2009). Solos marinhos da baixada santista – características e propriedades geotécnicas. Ed. Oficina de Textos.

Teixeira AH (1994). Fundações rasas na baixadas Santista. In A. Negro et al. (Eds), *Solos de Litoral de São Paulo* (pp. 137-154). Associação Brasileira de Mecânica dos Solos.

Wirth, X., Jiang, N-J., da Silva, T., Della Vechia, G., Evans, J., Romero, E., Bhatia, S. K. B. (2017). Undergraduate Geotechnical Engineering Education of the 21st Century. *Journal of Professional Issues in Engineering Education and Practice*, 143(3), 2516002.



## Appendix A – 2D Seepage Module

A1: *2DSeepage-Handout.pdf* – explains purpose of lab and outlines procedures to be followed

A2: *2DSeepage-Guidelines.pdf* – provides guidance on how to use Midas to conduct the analyses and extract the results

A3: *2DSeepage-Worksheet.pdf* – to be used to summarize data and provide answers

Additional files for download:

*2DSeepageLab-dam\_no\_screen.gts* – input file for Midas software

*2DSeepage-Spreadsheet.xlsx*

## Laboratory on 2D Seepage using FEM

### Purpose

- During this lab you will learn to:
  - 1) Use flow nets to determine important quantities related to flow and pressure fields in a porous medium.
  - 2) Gain exposure to the finite element method, a numerical method that is widely used in practice to solve many engineering problems.

### Equipment and Software

- Your laptop on which you will have loaded the Midas GTX NS Software (see document with installation instructions). Software download **MUST** be completed ahead of time.

### Reading

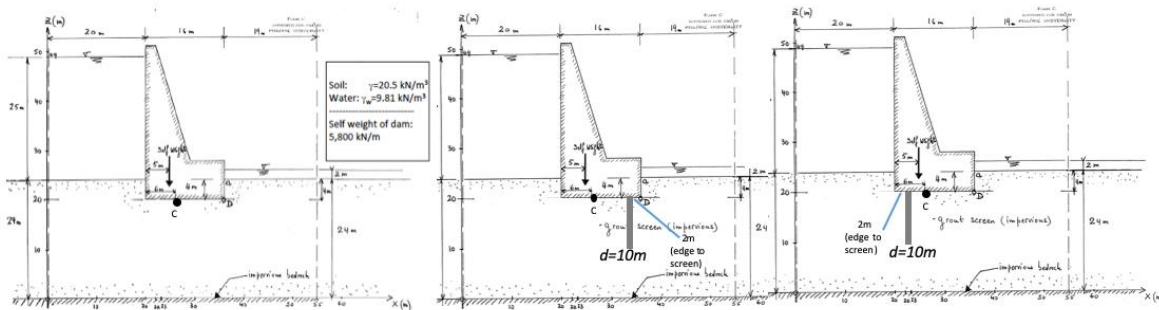
- Posted handouts and class notes in Blackboard. See also Section 7.7 in the HKS textbook.

### Handouts and files

- *2DSeepage-Handout.pdf* (this handout) – explains purpose of lab and outlines procedures to be followed
- *2DSeepage-Guidelines.pdf* – provides guidance on how to use Midas to conduct the analyses and extract the results (software specific)
- *2DSeepage-Worksheet.pdf* – to be used to summarize data and provide answers
- *2DSeepageLab-dam\_no\_screen.gts* – input file for Midas software
- *2DSeepage-Spreadsheet.xlsx* – excel file to be used to perform one of the required calculations

### Overview

The figures below present the geometry and material data of the problems to be analyzed.



The problems are two-dimensional and the flow conditions in the foundation soil are steady-state flow with isotropic permeability. The figures show a concrete dam, considered impervious, which is supported by a permeable foundation soil with hydraulic conductivity  $k=10^{-4}$  m/s. An impervious horizontal bedrock is present at fairly large depth. Each figure represents a different configuration for the same dam project: the first case has no grout screen, while the second and third cases involve construction of a 10 m deep grout screen, in one case 2 m from the upstream edge of the dam, in the second case 2 m from the downstream edge. The grout screen below the dam is a zone of soil treated by injection of a cement-based mix in order to decrease its permeability to almost zero. Typically, this grouting is performed from the surface prior to construction of the dam or from a gallery located at the base of the dam. In this lab, you will assume that the grouted zone is impervious. Note that the grout screen is not considered a structural element of the dam; its only purpose is to control the groundwater flow.

In this lab you are asked to:

- Perform manual calculations using a flow net provided to you for the 2D problem involving flow below the dam with no grout screen. You will calculate the flow rate, the maximum exit gradient, the pore water pressure at select points and the total uplift force on the dam.
- Use Midas GTS NX, a finite element program, to perform calculations to: establish the effects of the presence of the grout screen below the dam; and to define its optimal location and depth.

## Report

No report is required for this lab. You are required to complete the questionnaire by the end of lab and hand it in.

## Procedure:

### PART A – Analysis using flow nets

The figure on page 6 of this handout presents the geometry and material data of the first problem to be analyzed. Use the already drawn flow net to determine the pore pressure at Point C below the dam, the uplift force acting on the dam, the total seepage flow rate per unit length of dam (expressed in  $m^3/s/m$ ) and the maximum exit gradient. Summarize these values in the table at the top of [2DSeepage-Worksheet.pdf](#)

Note: there is no need for any equipotential to go through point C (you can interpolate values of the head linearly across each head drop).

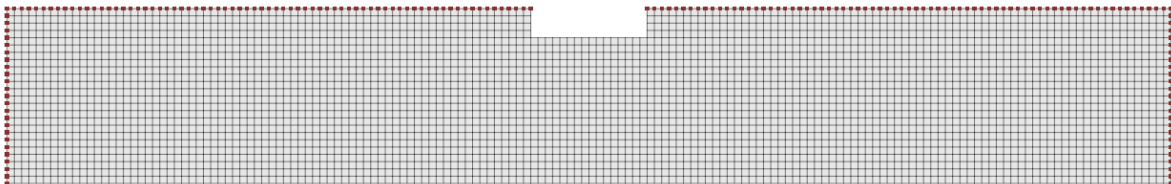
To calculate the uplift force on the dam you will need to determine the pore pressure along the base of the dam. The TAs will show you how to do this calculation in two different ways: one more accurate, the second more approximate but quicker.

### PART B – Analysis using Midas GTX NS – EFFECT OF CONSTRUCTING A GROUT SCREEN

In the second part of the lab you will use the Midas GTX NS Finite Element (FE) software to repeat the calculations done above and to explore the impact of constructing a grout screen under the dam. The finite element method is widely employed in civil engineering to solve complex structural and geotechnical engineering problems. When setting up a finite element analysis there are a few important decisions to make. The most important ones pertain to:

- the selection of the dimensions to be investigated, i.e. how far should I set the model boundaries?
- the discretization of this domain into “finite elements”, i.e. how many elements should I use? (note that in the same model you can have areas of finer mesh and areas of coarser mesh);
- the type of element(s) to use;
- the selection of the boundary conditions (constant head or impermeable boundaries);

For the purpose of this lab exercise these decisions have been made for you, and the resulting model for the dam without the grout screen has already been prepared for you, and is shown below.



This is the model that will appear once you start the problem and open the file (*2DSeepageLab-dam\_no\_screen.gts*) provided to you (see slide 2 of pdf entitled “*2DSeepage-Guidelines.pdf*”).

This same document provides step by step instructions on how to:

- perform the analysis (slide 4),
- visualize and extract the results (slides 5-12)
- modify the model to include the grout screen (slide 13)

The last slide explains how to recover a window when you inadvertently close it (slide 14).

You are expected to complete the following 3 analyses:

1. The **first analysis** (2D FLOW-DAM WITH NO GROUT SCREEN) pertains to the dam without a grout screen.

Follow the instructions in slides 2-4 to open the model and run the analysis.

- a) Follow the instructions on slides 5-7 to visualize the equipotential lines and learn how you would draw a flow net. The point here is *not* for you to reproduce the flow net (Midas provides a numerical solution – it does not need the flow net), but simply to see how you can visualize flow lines, and “sketch” a flow net. Answer the following questions in your worksheet:

B1 – What is an equipotential line?

B2 – What is a flow line?

B3 – What is the angle between equipotential lines and flow lines? Why?

B4 – How does the total head vary as a function of depth far to the left of the dam? What does this imply? What about far to the right?

- b) Follow the instructions on slide 8 to learn how to extract results (total head, pressure, gradient, etc.) at any location
- c) Follow the instructions on slide 9 to visualize the pore pressure contours and determine the pore pressure at point C. Add this value to the summary table in you worksheet
- d) Follow the instructions on slide 10 to determine the uplift force on the dam. You can use *2DSeepage-Spreadsheet.xlsx* to perform uplift calculations. Add this value to the summary table
- e) Follow the instructions on slide 11 to determine the maximum exit gradient Add this value to the summary table
- f) Follow the instructions on slide 12 to determine the total flow rate under the dam Add this value to the summary table
- g) Save the file.
- h) Compare the FE results to those obtained from the hand calculations using the flow net and answer question in your worksheet.

B5 – Compare the results from Midas with those calculated by hand using the flow net. Comment.

2. The **second analysis** (2D FLOW – EFFECT OF 10 m UPSTREAM GROUT SCREEN) pertains to the case with a **10 m deep grout screen placed under the upstream end of the dam – 2 m from the edge.**

- a) See slide 13 in *2DSeepage-Guidelines.pdf* for how to modify the model with a 10 m deep grout screen placed 2 m from the edge of the upstream end of the dam. Save the file again with a new name (e.g. *2DSeepageLab-dam\_10m upstream screen.gts*).
- b) Perform the analysis again (slide 4), and follow the instructions in slides 5-12 recording the same results in the summary table.
- c) Follow the instructions on slide 9 to visualize the pore pressure contours and determine the pore pressure at point C. Add this value to the summary table

B6 – What is the effect of the grout screen on the pressure at C? Explain.

- d) Follow the instructions on slide 10 to determine the uplift force on the dam. You can use *2DSeepage-Spreadsheet.xlsx* to perform uplift calculations. Add this value to the summary table

B7 – What is the effect of the grout screen on the uplift force? Explain.

- e) Follow the instructions on slide 11 to determine the maximum exit gradient. Add this value to the summary table

B8 – What is the effect of the grout screen on the exit gradient? Explain.

- f) Follow the instructions on slide 12 to determine the total flow rate under the dam. Add this value to the summary table

B9 – What is the effect of the grout screen on the flow rate? Explain.

- g) Save the file and answer the following questions.

B10 – What would you expect to observe if the upstream grout curtain extended to a greater depth?

B11 – What would you expect to observe if the upstream grout curtain extended all the way to the impervious bottom?

3. The **third analysis** (2D FLOW – EFFECT OF 10 m DOWNSTREAM GROUT SCREEN) pertains to the case with a **10 m deep grout screen placed under the downstream end of the dam – 2 m from the edge.**

- a) Generate the model for this analysis starting from the original model with no grout curtain. Save the file again with a new name (e.g. *2DSeepageLab-dam\_10m downstream screen.gts*).
- b) Perform the analysis again (slide 4), and follow the instructions in slides 5-12 recording the same results in the summary table.
- c) Follow the instructions on slide 9 to visualize the pore pressure contours and determine the pore pressure at point C. Add this value to the summary table

B12 – What is the effect of the grout screen on the pressure at C? Explain.

- d) Follow the instructions on slide 10 to determine the uplift force on the dam. You can use *2DSeepage-Spreadsheet.xlsx* to perform uplift calculations. Add this value to the summary table

B13 – What is the effect of the grout screen on the uplift force? Explain.

- e) Follow the instructions on slide 11 to determine the maximum exit gradient. Add this value to the summary table

B14 – What is the effect of the grout screen on the exit gradient? Explain.

- f) Follow the instructions on slide 12 to determine the total flow rate under the dam. Add this value to the summary table

B15 – What is the effect of the grout screen on the flow rate? Explain.

- g) Save the file and answer the following question.

B16 – What would you expect to observe if the upstream grout curtain extended to a greater depth?

## **PART C – Analysis using Midas GTX NS – DESIGNING THE GROUT SCREEN**

The dam design needs to satisfy two design requirements:

Requirement 1: the **maximum exit gradient** < 0.5

Requirement 2: the **uplift force (in kN/m) x 2.5** < **self weight of the dam (in kN/m)**, i.e. less than 5800 kN/m (that is a factor of safety of 2.5).

- a) Start by answering question C1 on your worksheet which examines the three scenarios investigated to this point. You should find that in none of the cases the requirements are satisfied. For the two solutions with a grout screen consider a deeper screen. While the idea would be to find the minimum depth of the screen that satisfies both criteria, simply increase the screen depth to 15 m.

C1 – Based on the results obtained using Midas, do the three design solutions satisfy?

- b) Repeat the analyses with a 15 m curtain upstream and a 15 m curtain downstream and report the results in the table and answer questions C2-C6.

C2 – With a deeper curtain, do the design solutions satisfy?

C3 – How can you explain these results?

C4 – What design would you recommend?

C5 – What additional analysis should be performed? Why?

# PORE PRESSURE AT C

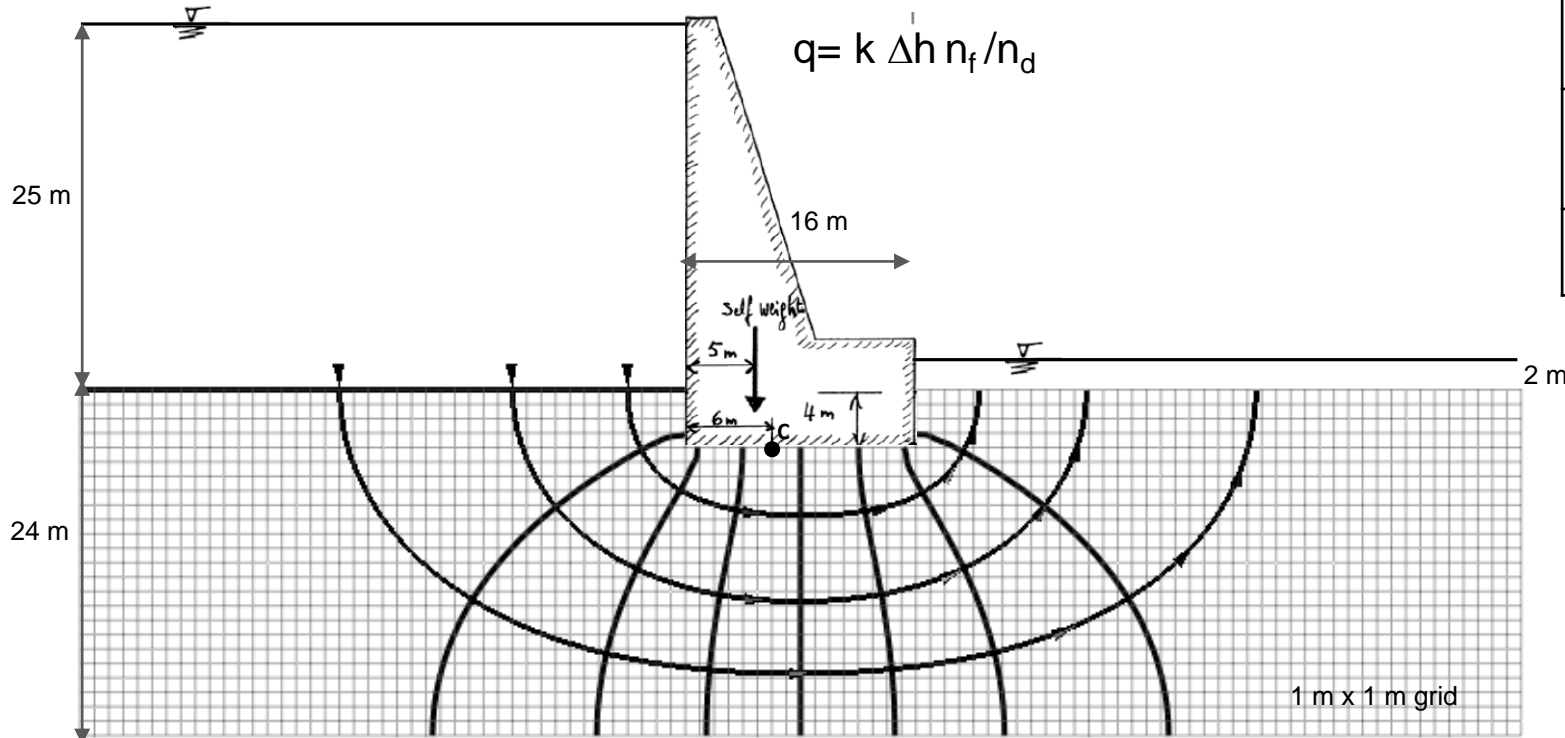
# MAXIMUM EXIT GRADIENT

# UPLIFT FORCE

$$i_{max} =$$

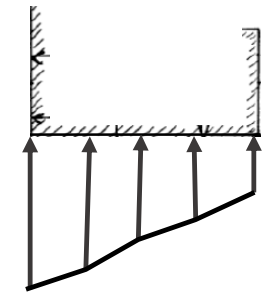
# FLOW RATE UNDER THE DAM

$$q = k \Delta h n_f / n_d$$

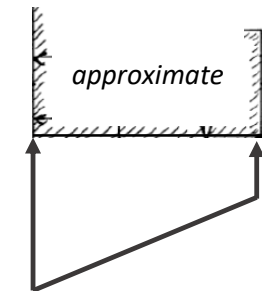


Uplift Force Calculations	Pressure head (m)	Pore pressure (kPa)	Portion of dam base (m)	Uplift force kN/m
Left corner			-	-
Equipotential line 4	17.5	171.7	4	
Equipotential line 5	14.5	143.5	4	
Equipotential line 6	11.8	115.8		
Right corner	10.3	101.2		

Soil:  $k=10^{-4}$  m/s  
 $\gamma = 20.5$  kN/m<sup>3</sup>  
 Self weight of dam:  
 5,800 kN/m



TOTAL=



TOTAL=  
 approximate



# Laboratory on 2D Seepage using FEM

Tutorial for conducting the analysis using the Midas GTS NX software

Prepared by O. Vitali, A. Getchell and M. Santagata – Spring 2020

*to be used in conjunction with Lab Master Handout*

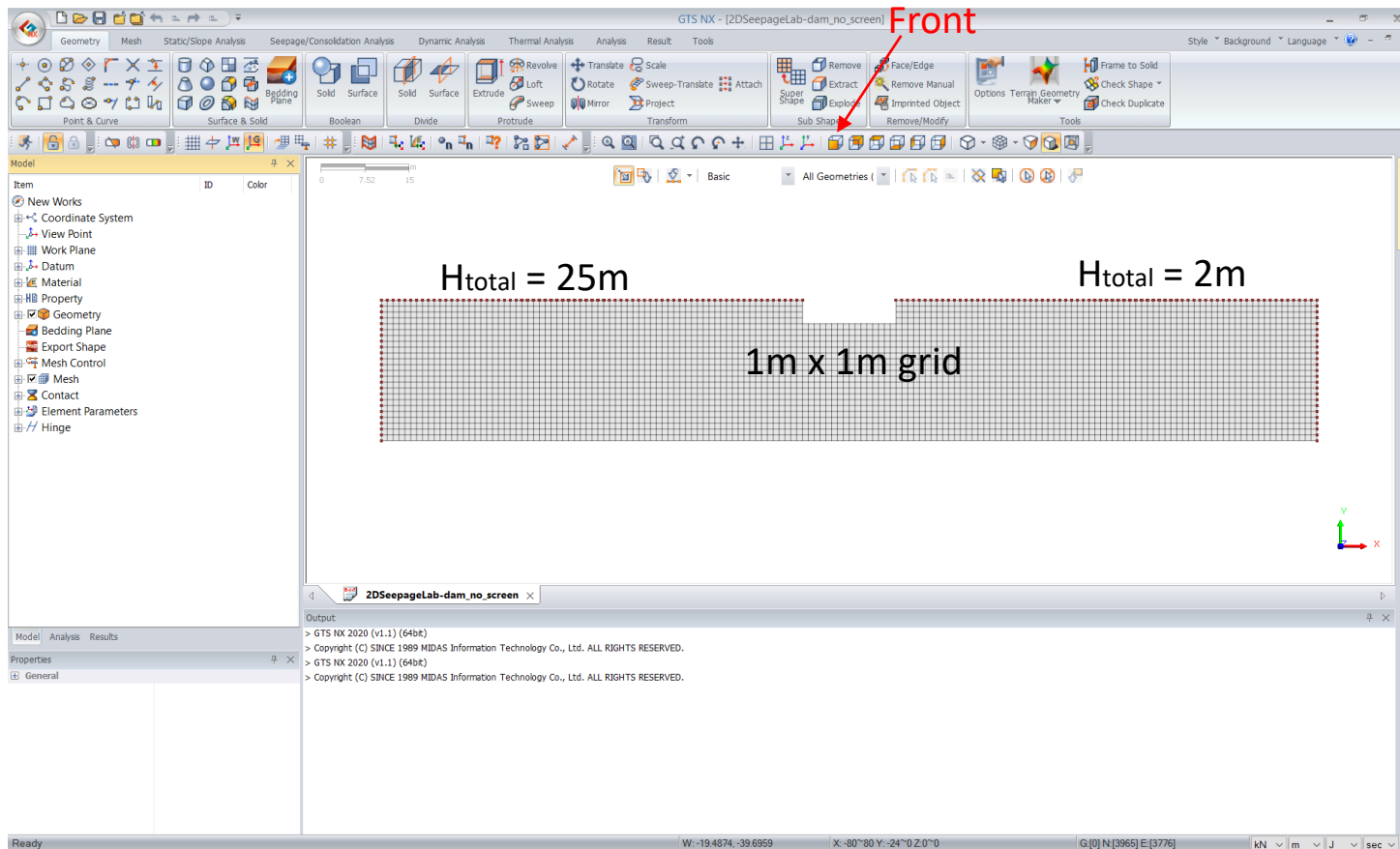
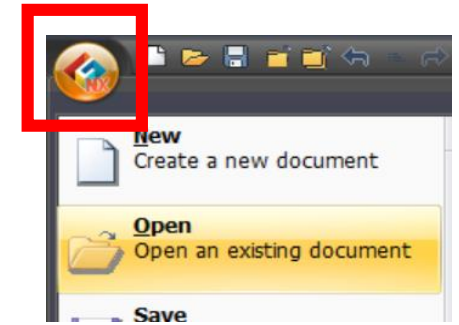


# STEP 1: GETTING STARTED

## 1a. Loading the model

Open 2DSeepageLab-dam\_no\_screen.gts (double click on the file or go to File → Open and select file)

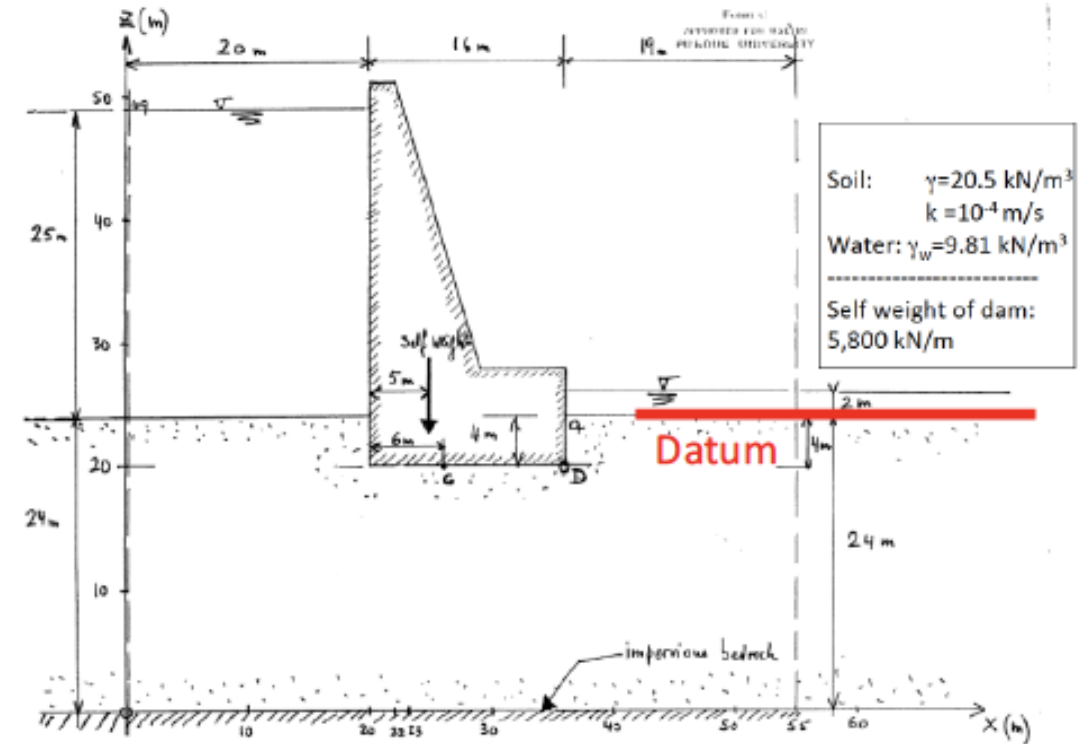
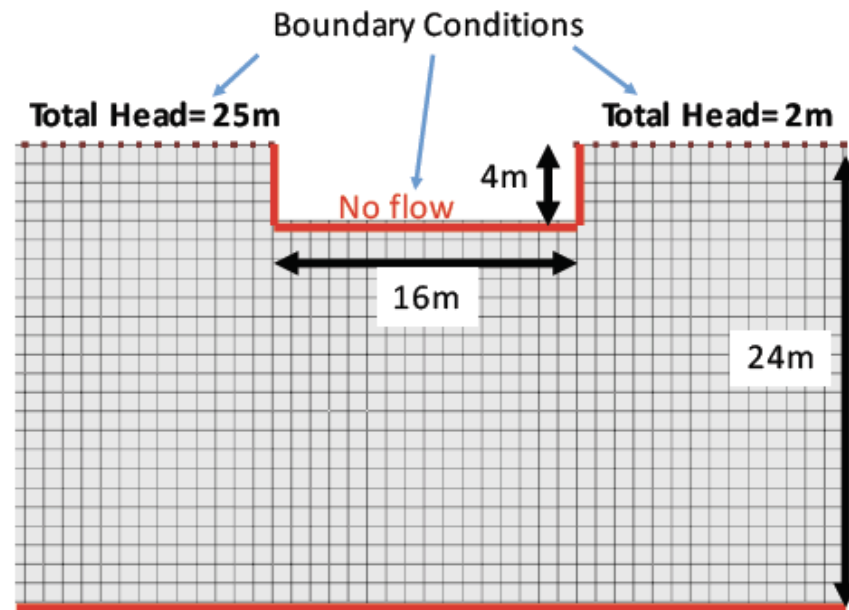
File



This is how the model will look.  
You may use the cursor to zoom in and zoom out.  
Hint: Click on “Front” to center the model.

# STEP 1: GETTING STARTED

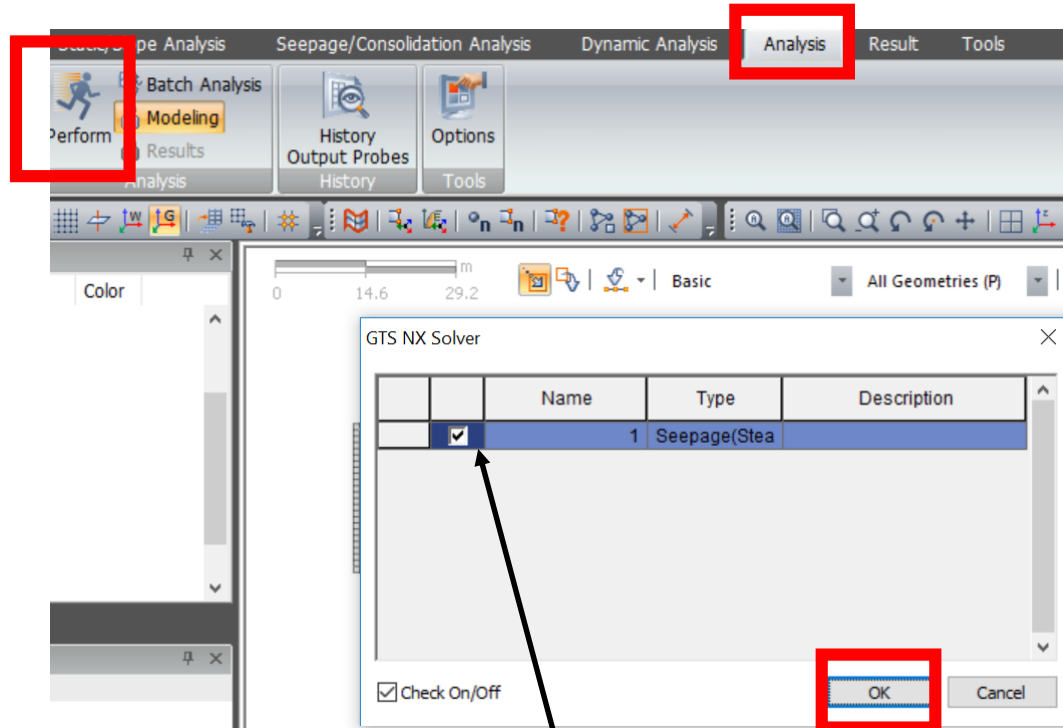
## 1a. Inspecting the model: FEM mesh, boundary conditions, and materials



The finite element model has the same geometry and boundary conditions of the dam you analyzed using a flow net. In the model the datum is placed at the top of the soil. The finite element mesh is a 1m x 1m rectangular grid.

## STEP 2: PERFORMING THE ANALYSIS

To run the model, go to “Analysis” and click on “Perform” and then “Ok”. The program will calculate the solution.



Make sure this box is checked before clicking OK.

Be patient as the model runs.

# STEP 3: ANALYZING THE RESULTS

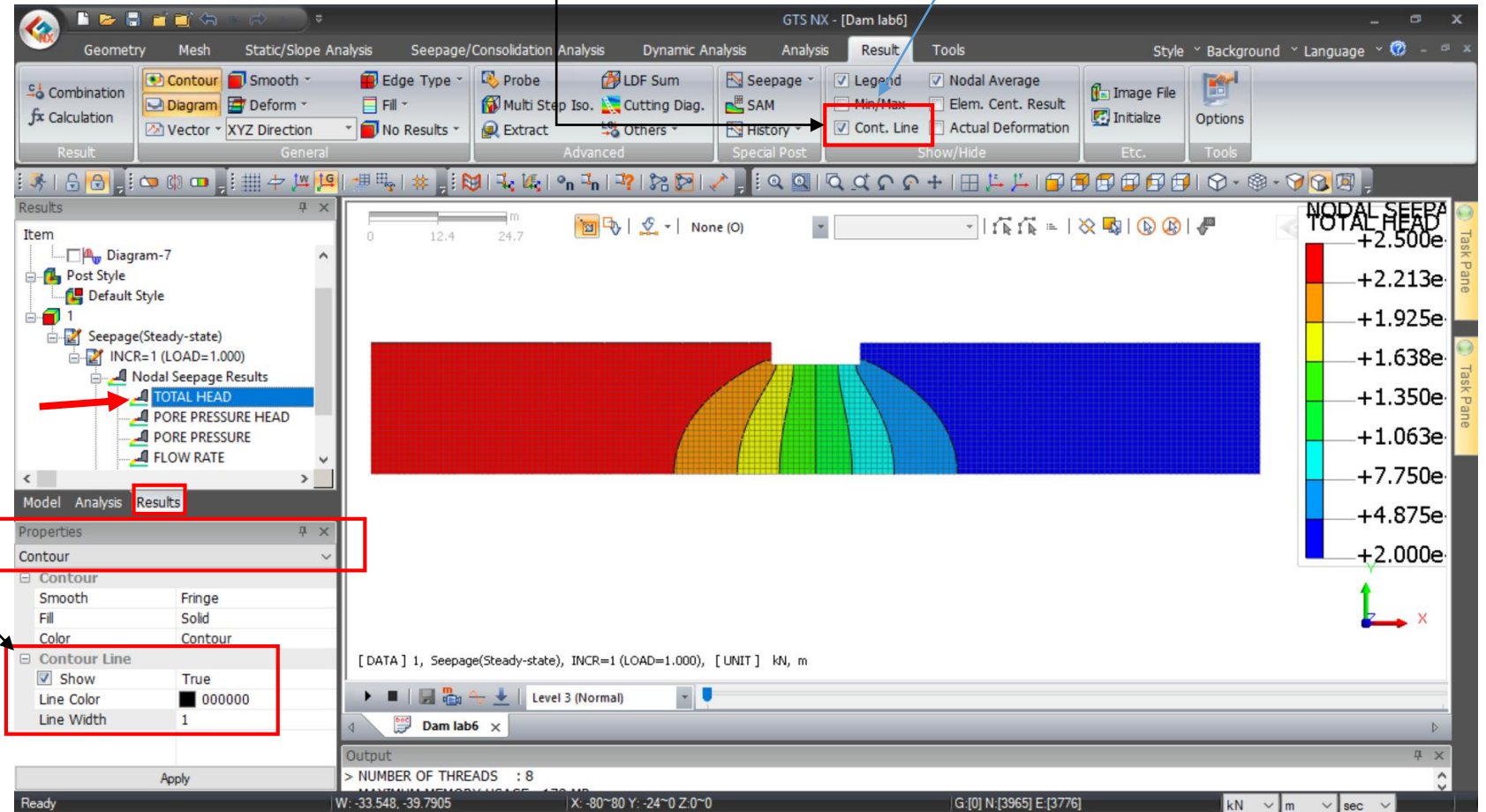
## 3a. Visualizing equipotential lines

- Go to Results → Seepage → Nodal Seepage Results → TOTAL HEAD (you must double click)
- Check “Cont. Line” to see the Equipotential Lines.
- To edit Contour lines, go to Properties – Contour

*To see contour lines make sure this box is checked*

*To edit contour lines*

You can plot the contours of other parameters including pressure head and pore pressure.



## STEP 3: ANALYZING THE RESULTS

### 3a. (cont.) Visualizing equipotential lines

To change the number of equipotential lines, go to: Properties → Legend → Number of Bands

The screenshot displays the GTS NX software interface for a dam analysis. The main window shows a contour plot of the total head distribution across a cross-section of a dam. The plot is color-coded, with red representing the highest head values and blue representing the lowest. A legend on the right side of the plot lists the head values in meters, ranging from +2.000e to +2.500e. The Properties dialog box is open, showing the Legend tab. The 'Number of Bands' is set to 16. A red arrow points from the text 'Number of Equipotential Drops' to the 'Number of Bands' field.

*Number of Equipotential Drops*

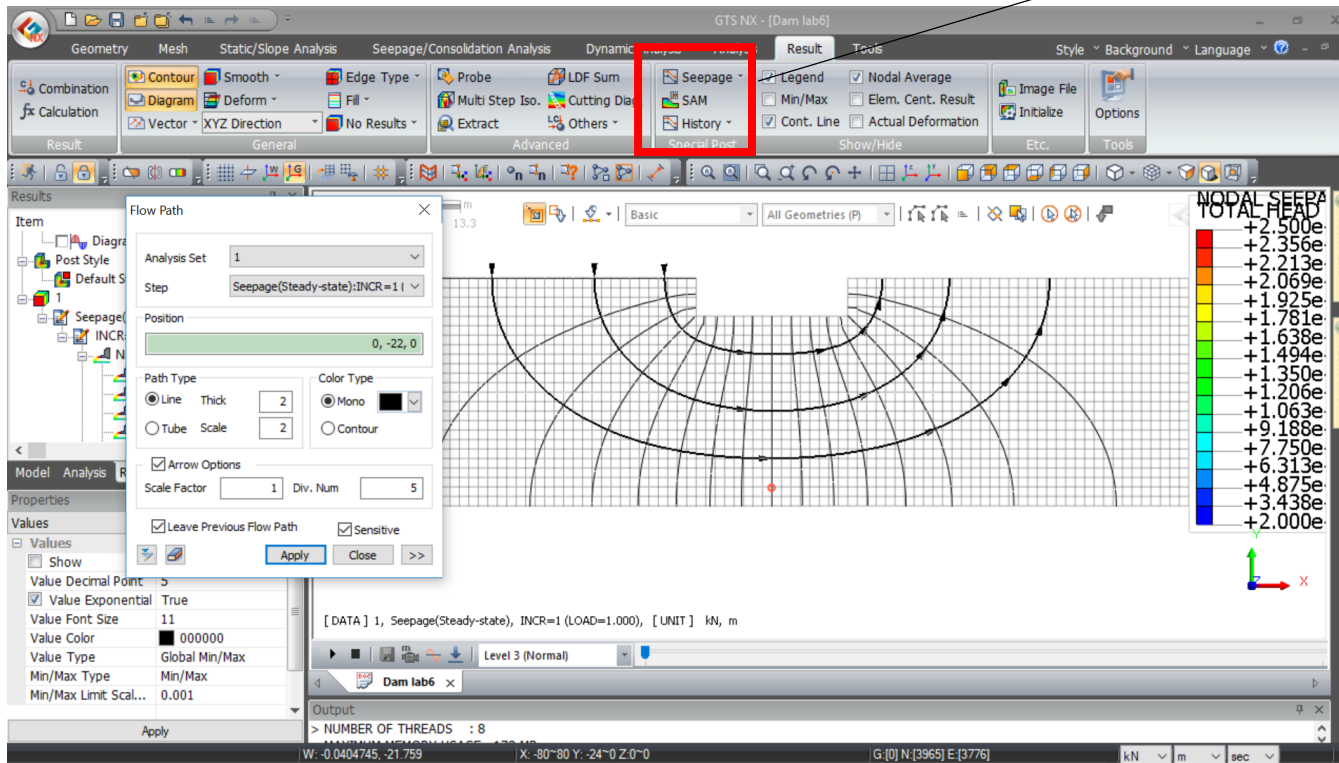
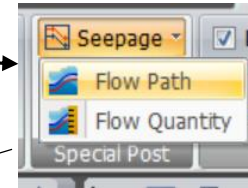
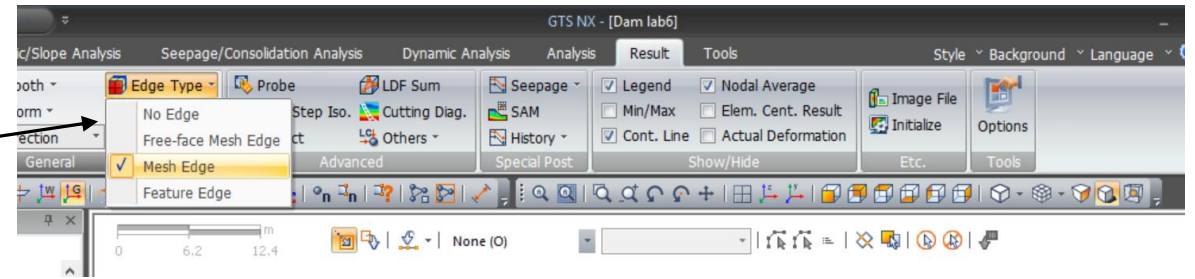
Legend Value (m)
+2.500e
+2.356e
+2.213e
+2.069e
+1.925e
+1.781e
+1.638e
+1.494e
+1.350e
+1.206e
+1.063e
+9.188e
+7.750e
+6.313e
+4.875e
+3.438e
+2.000e

# STEP 3: ANALYZING THE RESULTS

## 3b. Visualizing flow paths and building a flow net

- First, make sure the grid is visible (Results → Edge Type → Mesh Edge)
- Second, visualize Total Head with Contour
- Third, go to Results → Seepage → Flow Path.

Click on any node to see the flow path going through it.



### Notes:

You cannot delete a flow line once you draw it (you have to delete all), so be careful.

Start in a place where it is easy to figure out the position that will create a square mesh. In the flow net on the left this is right below the center of the dam.

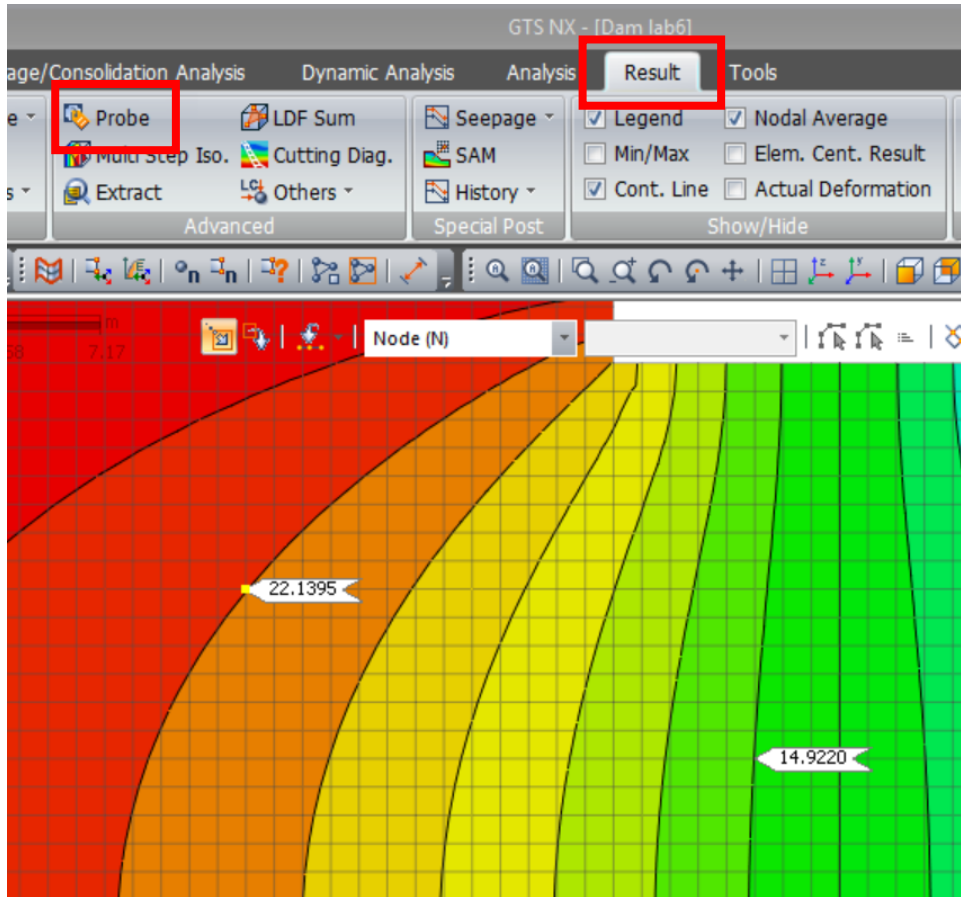
You can also decide to add equipotentials while you are putting in flow lines. The result should be no different than a flow net you draw by hand following all the rules. Before moving to the next step, you may want to take a screen shot of the flow net.

Close the flow path window.

## STEP 3: ANALYZING THE RESULTS

### 3c. Extracting the results at the nodes to obtain pore pressures and gradients

- Visualize the contours of the property you are interested in getting values for.  
Go to Results → Seepage → Nodal Seepage Results → DESIRED PROPERTY (e.g. total head)
- Go to: Results → Probes → Click on the desired node (intersection of the grid).



The value will appear on the contours (as shown on the left).  
It will also be saved in a table.

To visualize the data in tabular form, hold the mouse on Probe Results. Values can be copied directly from this table and pasted in excel to perform calculations.

The 'Probe Results' dialog box is shown. The 'Node' radio button is selected. The 'Results' table shows two rows of data:

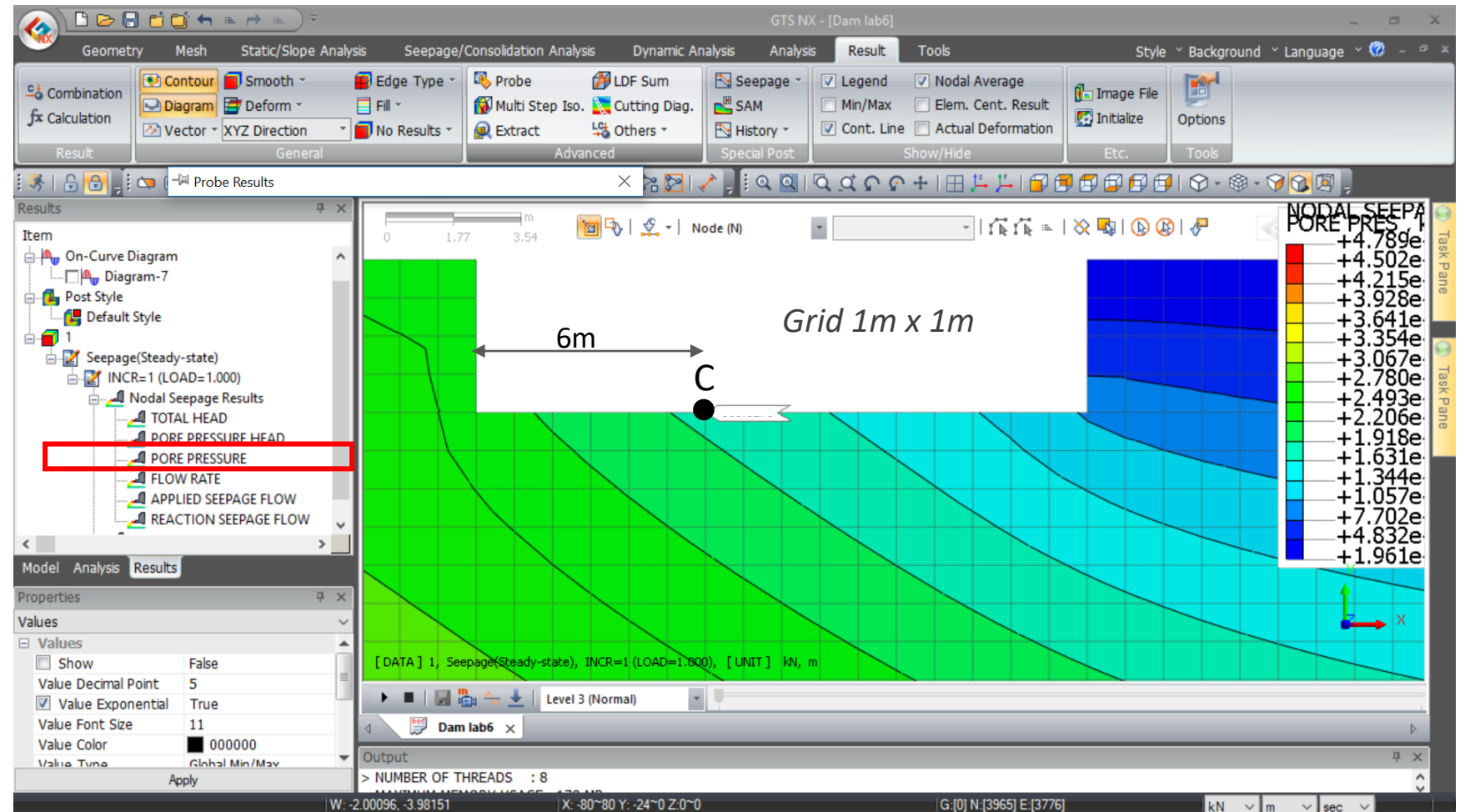
Show	Type	ID	Value
<input checked="" type="checkbox"/>	Node	1470	14.9220
<input checked="" type="checkbox"/>	Node	2436	22.1395

# STEP 3: ANALYZING THE RESULTS

## 3c. (cont.) Extracting the results at the nodes

To get the *pore pressure at point C*

- Go to: Results → Nodal Seepage Results → Pore Pressure. Then use the Probe tool to get the pore pressure at Point C





## STEP 3: ANALYZING THE RESULTS

### 3c. (cont.) Extracting the results at the nodes

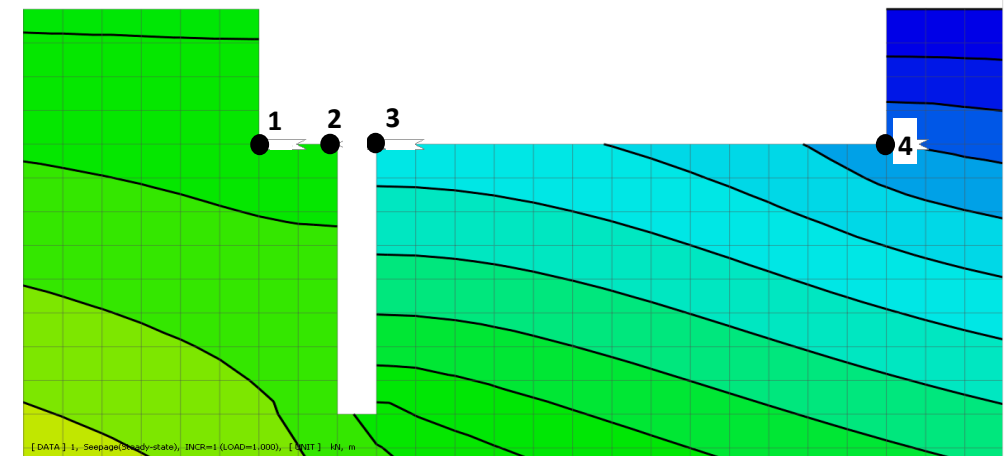
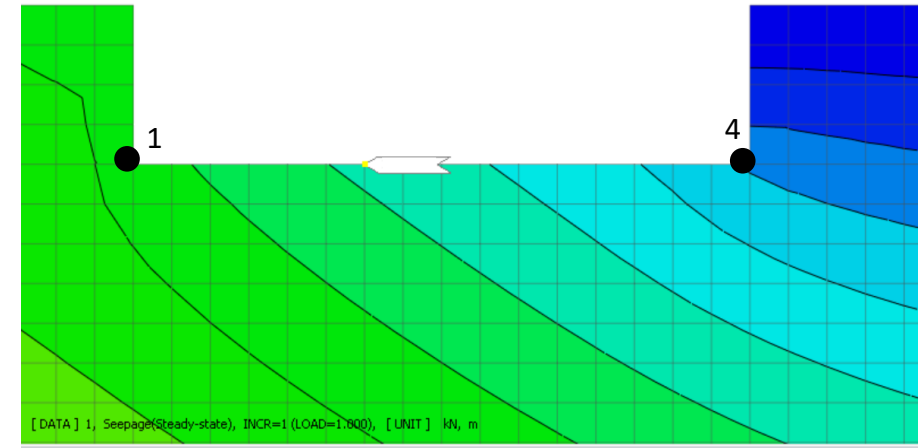
To get the **pore pressure at the base of the dam** for calculation of the uplift force:

#### - **Case with no grout screen (base case)**

Use the Probe tool to get the pore pressure at the two edges of the dam (see top figure). Copy the values in the excel file prepared for you to calculate the uplift force. You may use either sheet in the file. Insert the values in the cells under “Pore pressure 1” and Pore pressure 4,” on the row for zero depth of the grout screen. The sheet automatically calculates the uplift force (column H).

#### - **Case with grout screen**

Use the Probe tool to get the pore pressure at the four points identified in the bottom figure. Copy the values in the excel file prepared for you to calculate the uplift force. Use the sheet corresponding to the correct position of the grout screen. Insert the values in the cells under “Pore pressure 1,” “Pore pressure 2,” “Pore pressure 3” and “Pore pressure 4” in the row for the appropriate depth  $d$  of the screen. The sheet automatically calculates the uplift force (column H).



*Note: this calculation neglects the contribution to uplift due to water pushing on the bottom of screen*

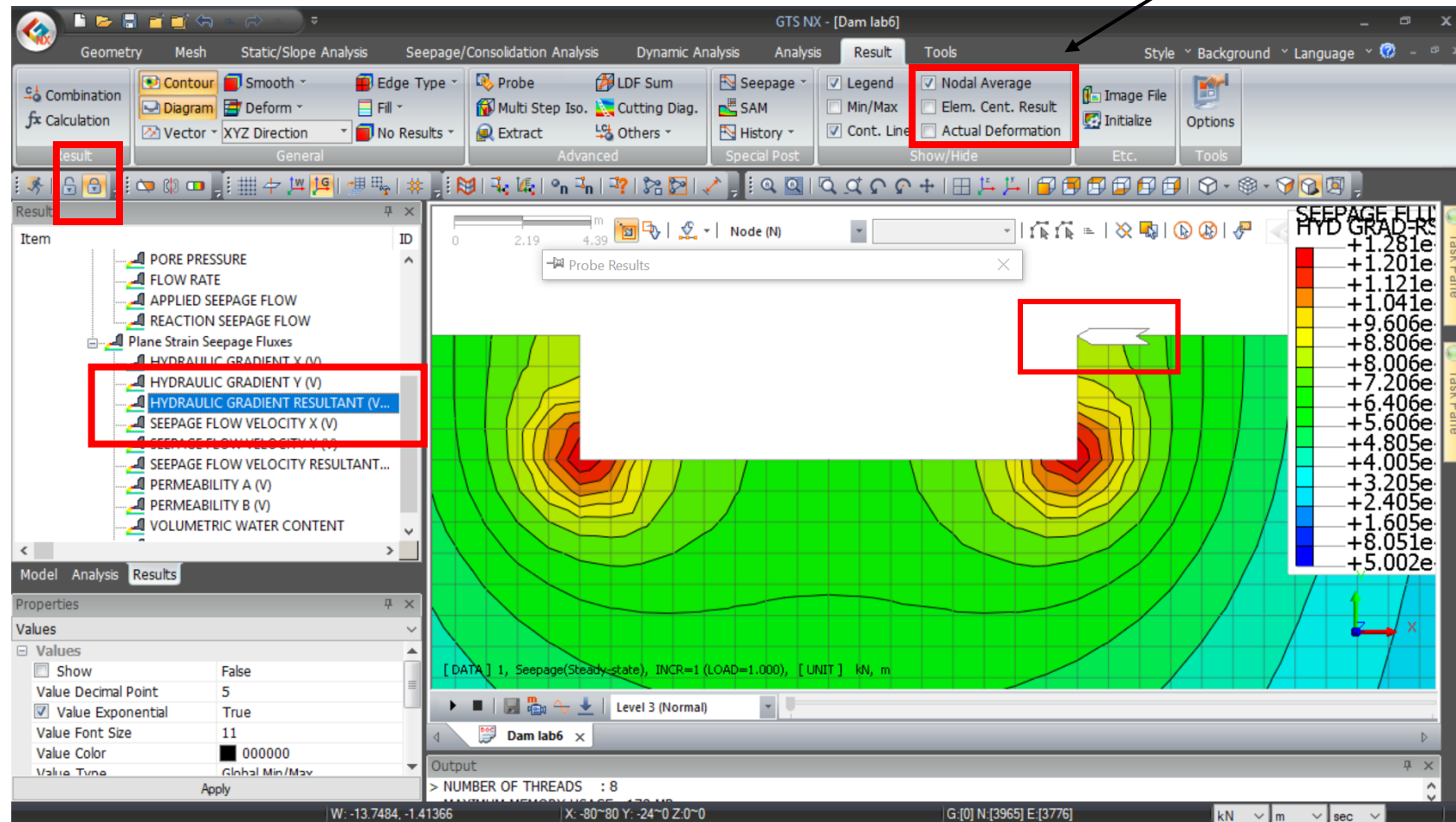
## STEP 3: ANALYZING THE RESULTS

### 3c. (cont.) Extracting the results at the nodes

#### To obtain maximum exit hydraulic gradient

- Go to: Results → Plane Strain Seepage Fluxes → Hydraulic Gradient Resultant (this is the 3<sup>rd</sup> option)
- Use the Probe tool to get the maximum exit gradient.

Make sure "nodal average" is selected



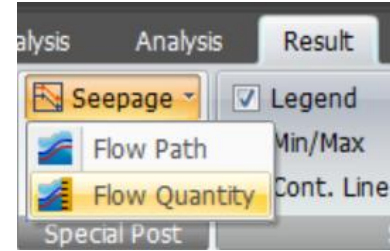
# STEP 3: ANALYZING THE RESULTS

## 3d. Obtaining flow rates

First: Go to Result → Seepage → Flow Quantity

Second: Click Add -- select the nodes shown below – select a name (e.g. 1) – click OK

Third: Check “1”– Click Calculate



Flow Quantity ×

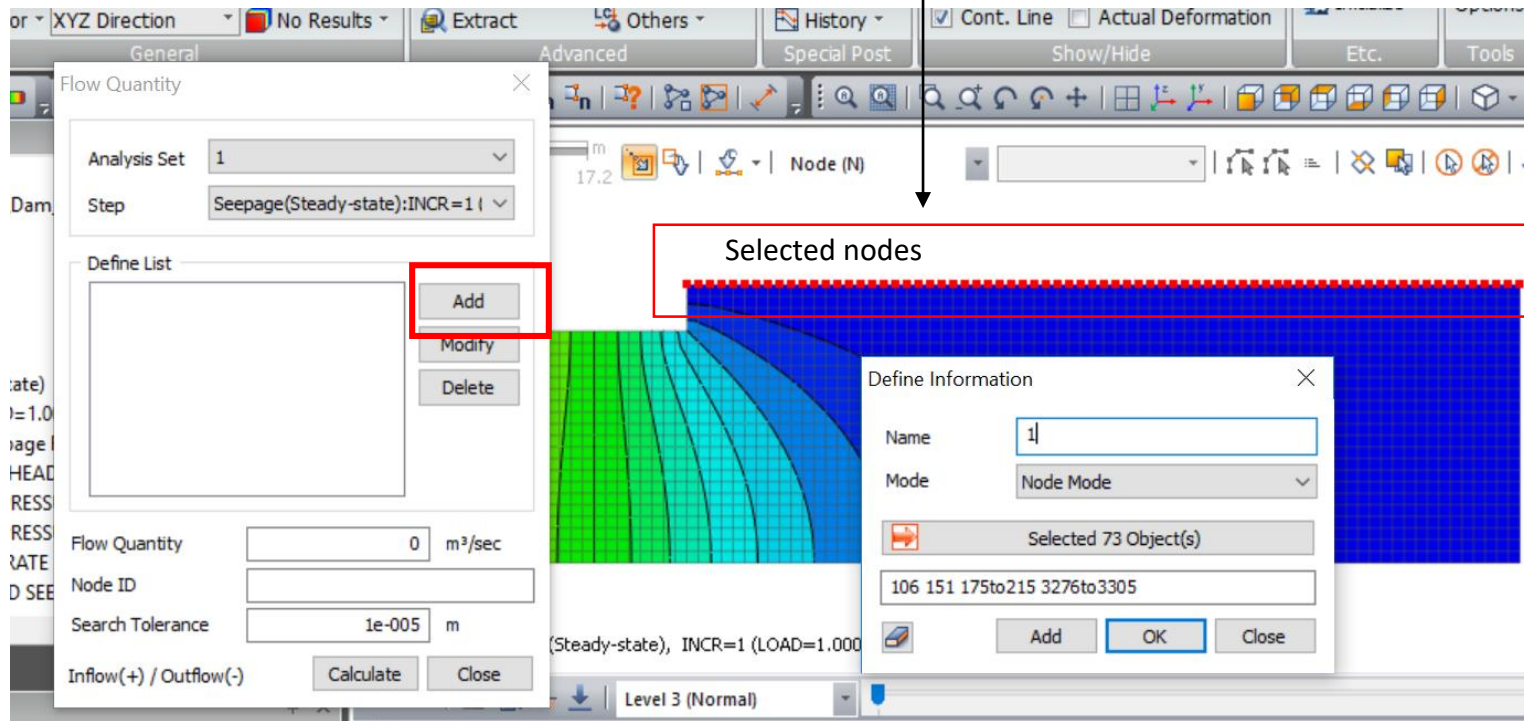
Analysis Set: 1  
Step: Seepage(Steady-state):INCR=1

Define List

<input checked="" type="checkbox"/>	1
-------------------------------------	---

Add  
Modify  
Delete

Flow Quantity:  m<sup>3</sup>/sec  
Node ID:   
Search Tolerance:  m  
Inflow(+) / Outflow(-):



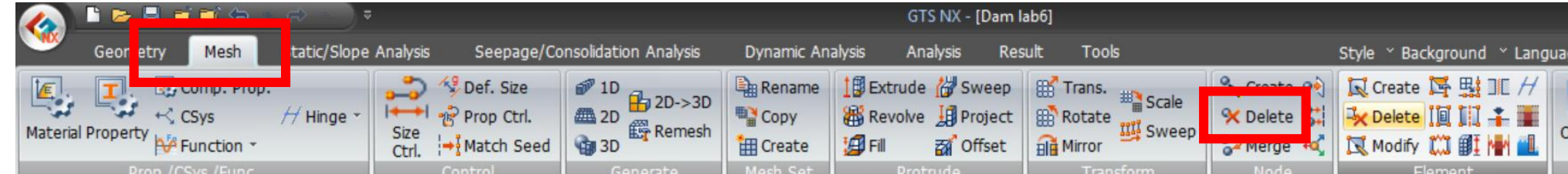
## STEP 4: MODIFYING THE MODEL

To conduct the analyses for the other geometries required for this assignment you will need to modify the model to include the grout curtain.

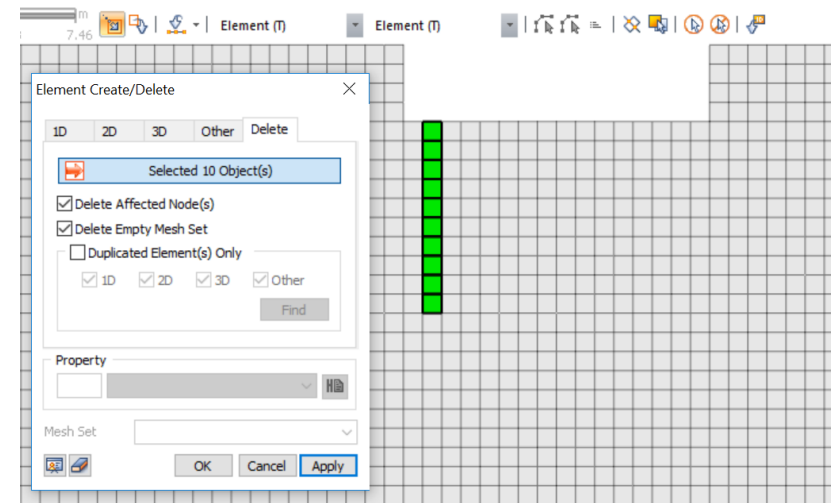
The new model should be built starting from 2DSeepageLab-dam\_no\_screen.gts by deleting elements (squares of the grid).

To delete an element; 1) Click on Pre-mode (open lock figure)

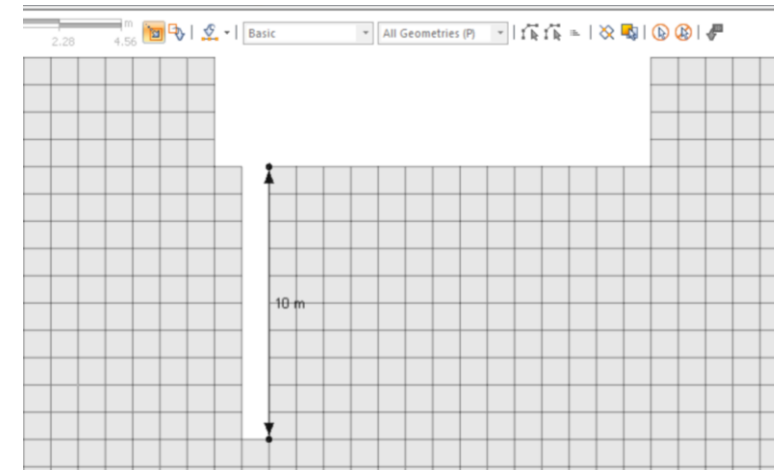
2) go to Mesh → Delete Elements



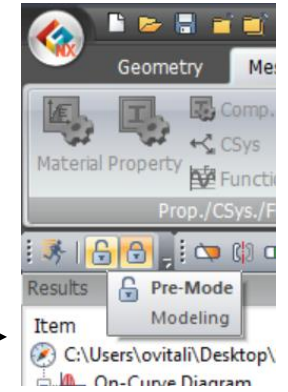
3) Select the elements you want to delete and click OK



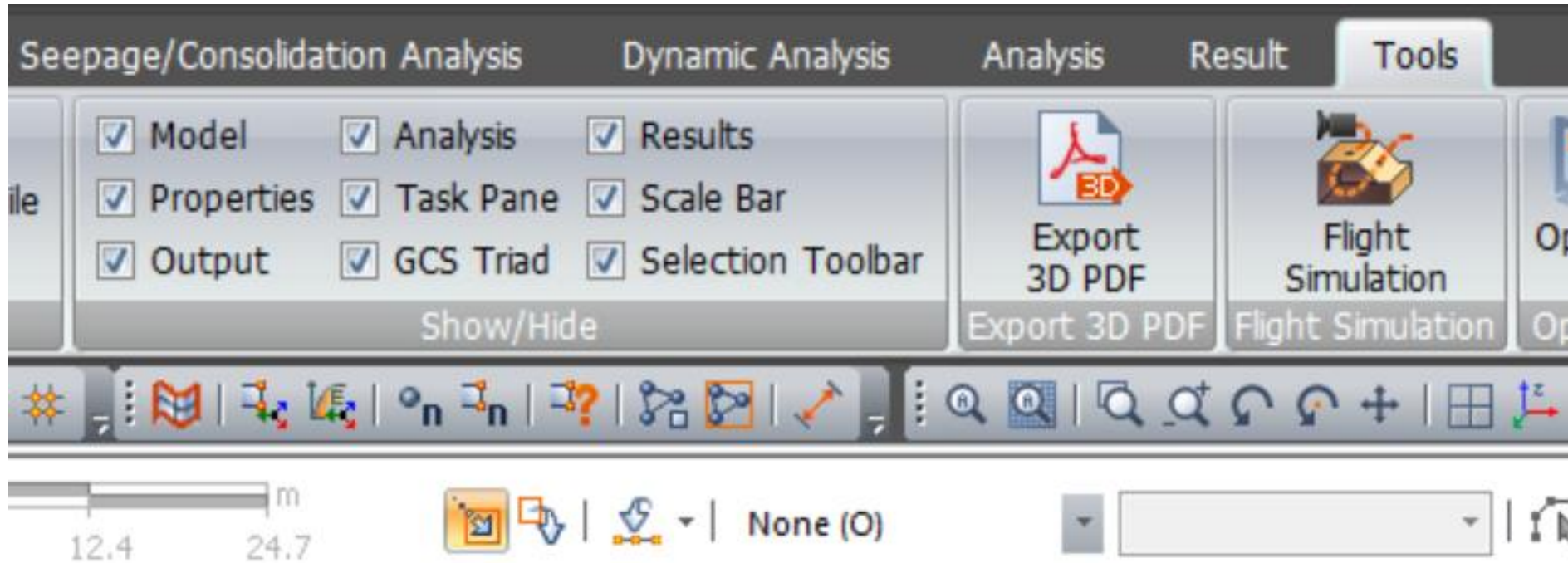
4) Save the new geometry with another name using "Save as."



Run the model again (see slide 3) to get the results.



FINAL HINT: If you close one of the “working windows” (probably by mistake), you can reopen it in Tools.



**Laboratory on 2D Seepage using FEM  
 WORKSHEET TO BE TURNED IN AT THE END OF LAB**

**SUMMARY TABLE:**

	<i>Pressure at C (kPa)</i>	<i>Uplift force (kN/m)</i>	<i>Exit gradient</i>	<i>Flow rate under the dam (m<sup>3</sup>/s/m)</i>
<i>No screen – Flow net results (PART A)</i>				
<i>No screen – Midas (PART B)</i>				
<i>d=10 m upstream screen – Midas (PART B)</i>				
<i>d=10 m downstream screen – Midas (PART B)</i>				
<i>d=15 m upstream screen – Midas (PART C)</i>				
<i>d=15 m downstream screen – Midas (PART C)</i>				

**2D FLOW – DAM WITH NO GROUT SCREEN (PART B)**

B1) What is an equipotential line? .....

B2) What is a flow line? .....

B3) What is the angle between equipotential lines and flow lines? .....

Why? .....

.....

B4) How does the total head vary as a function of depth far to the left of the dam? What does this imply? What about far to the right?

.....

.....

B5) Compare the results from Midas with those calculated by hand using the flow net.

*Pressure at C (% difference)*..... *Uplift force (% difference)*.....

*Flow rate (% difference)*..... *i<sub>max</sub> (% difference)*.....

*Comments:*.....

.....

**2D FLOW – EFFECT OF 10 m UPSTREAM GROUT SCREEN (PART B)**

B6) What is the effect of the grout screen on the pressure at C? Explain.

.....  
.....

B7) What is the effect of the grout screen on the uplift force? Explain.

.....  
.....

B8) What is the effect of the grout screen on the exit gradient? Explain.

.....  
.....

B9) What is the effect of the grout screen on the flow rate? Explain.

.....  
.....

B10) What would you expect to observe if the upstream grout curtain extended to a greater depth, in terms of: NO CALCULATIONS REQUIRED (goes up/goes down/the same)

i) *Exit gradient:* ..... ii) *Flow rate:* ..... iii) *Uplift force:* .....

B11) What would you expect to observe if the upstream grout curtain extended all the way to the impervious bottom? NO CALCULATIONS REQUIRED – you may check with Midas

.....

**2D FLOW – EFFECT OF 10 m DOWNSTREAM GROUT SCREEN (PART B)**

B12) What is the effect of the downstream grout screen on the pressure at C? Explain.

.....  
.....

B13) What is the effect of the downstream grout screen on the uplift force? Explain.

.....  
.....

B14) What is the effect of the downstream grout screen on the exit gradient? Explain.

.....  
.....

B15) What is the effect of the downstream grout screen on the flow rate? Explain.

.....  
 .....

B16) What would you expect to observe if the downstream grout curtain extended to a greater depth, in terms of: NO CALCULATIONS REQUIRED (goes up/goes down/the same)

i) Exit gradient: ..... ii) Flow rate: ..... iii) Uplift force: .....

**DESIGN OF THE GROUT SCREEN (PART C)**

C1) Based on the results obtained using Midas, do the three design solutions satisfy (YES/NO)

	<i>No screen</i>	<i>10 m upstream screen</i>	<i>10 m downstream screen</i>
<i>1<sup>st</sup> requirement (<math>i_{max} &lt; 0.50</math>)?</i>	.....	.....	.....
<i>2<sup>nd</sup> requirement (uplift force x 2.5 &lt; 5,800kN/m)?</i>	.....	.....	.....

C2) With a deeper curtain, do the design solutions satisfy (YES/NO)

	<i>15 m upstream screen</i>	<i>15 m downstream screen</i>
<i>1<sup>st</sup> requirement (<math>i_{max} &lt; 0.50</math>)?</i>	.....	.....
<i>2<sup>nd</sup> requirement (uplift force x 2.5 &lt; 5,800kN/m)?</i>	.....	.....

C3) How can you explain these results?

.....  
 .....

C4) What design would you recommend?

.....

C5) What additional analyses should be performed? Why?

.....  
 .....



## **Appendix B – Stress Analysis Module**

B1: *StressAnalysis-Handout.pdf* – explains purpose of lab and outlines procedures to be followed

B2: *StressAnalysis-Guidelines.pdf* – provides guidance on how to use Midas to conduct the analyses and extract the results

B3: *StressAnalysis-Worksheet.pdf* – to be used to summarize data and provide answers

Additional files for download:

*StressAnalysisLab-model.gts* – input file for Midas software

*StressAnalysis-Spreadsheet.xlsx*

## Laboratory on Stress Analysis using FEM

### Purpose

- During this lab you will:
  - 1) Use the Midas GTS NX Finite Element software to perform stress analysis.
  - 2) Visualize and evaluate the distribution of stresses in the ground (i.e. stress bulbs and stress bulbs superposition) produced under different scenarios;
  - 3) Extract the stresses at specific points, and make comparisons to the results obtained using elastic solutions in Lab #7;
  - 4) Extract horizontal and vertical stress diagrams, plot graphs and evaluate the results.

### Equipment and Software

- Your laptop on which you will have loaded the Midas GTX NS Software (see document with installation instructions). Software download **MUST** be completed ahead of time.

### Reading

- Posted handouts and lab notes posted in Blackboard. See also Section 10.3 in the HKS textbook.

### Handouts and files

- *StressAnalysis-Handout.pdf* (this handout) – explains purpose of lab and outlines procedures to be followed
- *StressAnalysis-Guidelines.pdf* – provides guidance on how to use Midas to conduct the analyses and extract the results (software specific)
- *StressAnalysis-Worksheet.pdf* – to be used to summarize data and provide answers
- *StressAnalysisLab-model.gts* – input file for Midas software
- *StressAnalysis-Spreadsheet.xlsx* – excel file to be used to perform one of the required calculations

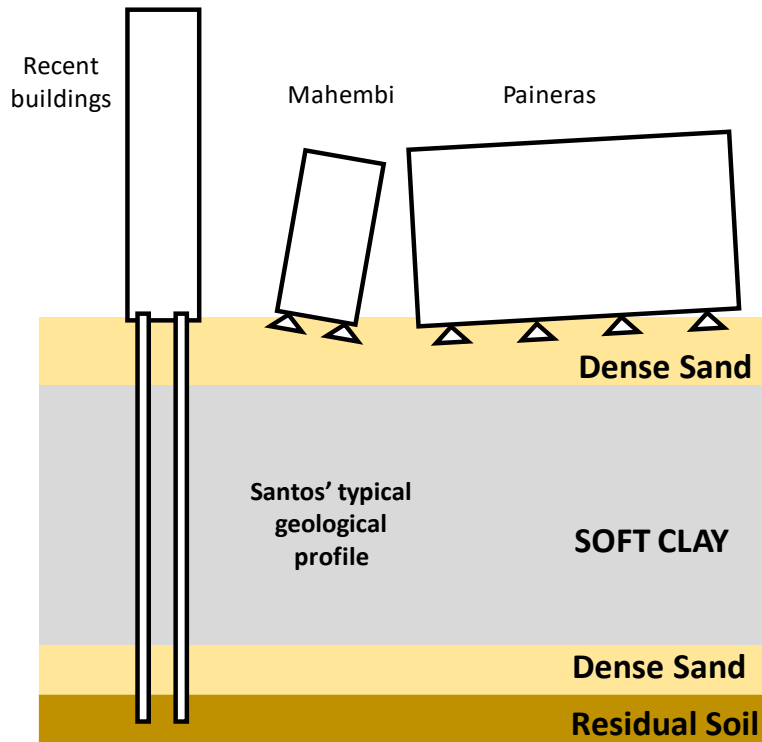
### Overview

The elastic solutions used in Lab #7<sup>1</sup> assume isotropic elastic behavior and homogenous soil conditions that is constant elastic properties ( $E$ ,  $\nu$ ). However, soils usually are heterogeneous and the stratigraphy at a given site typically involves layers with different properties. In these cases, elastic solutions may not be accurate. Additionally, the analytical solutions we examined are for simple loading conditions, where actual conditions in the field may be more complex. For more complex scenarios, where analytical solutions are not available, the Finite Element Method (FEM) is used. The FEM is a numerical method to solve Partial Differential Equations. Today, many professional FEM software packages, like Midas GTS NX, are available and are widely used in engineering practice for design. While more complex material models can also be incorporated in a finite element code, in this lab we will continue to rely on isotropic elasticity. Note that this is acceptable for calculating stresses but a more advanced model for soil behavior is required to derive accurate predictions of strains and settlements.

In this lab, we will analyze a tilted building in Santos, Brazil. A simplified geological profile of the city of Santos is shown in Figure 1. A layer of dense sand overlies a thick soft clay layer, under which lies another layer of dense sand. Because the superficial layer of sand has good mechanical properties, many tall buildings (up to 17 floors) were constructed in Santos between 1950 and 1970 on shallow foundations. This is the case of the Mahembi and Paineiras buildings shown in Figure 2. At that time, there was no practical technology to construct deep foundations, as is instead possible today (see leftmost building in Figure 1). Geotechnical engineers warned builders that because of the presence of a thick soft clay layer under the sand, large settlements and potential leaning of the buildings might occur over time. However, these warnings were ignored, and many buildings from that time period have experienced large settlements and are visibly tilted.

---

<sup>1</sup> Previous laboratory session in which the Boussinesq elastic solution was used to calculate the vertical stress increments for the same buildings



**Figure 1.** Typical geological profile of Santos, Brazil. Tilted old buildings constructed over shallow foundations and recent buildings over deep foundations.

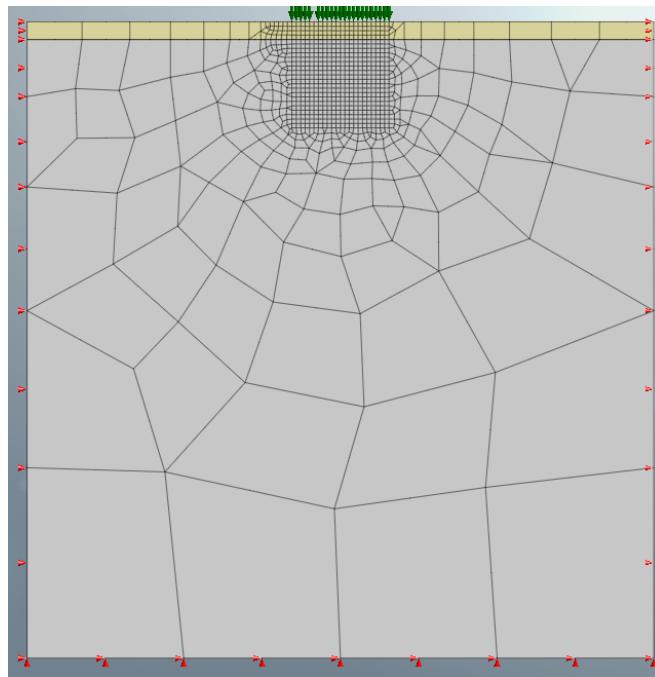


**Figure 2.** Photos of leaning buildings in Santos (on the left Mahembi building; on the right both Mahembi and Paineiras buildings).

In this lab you will assess the vertical stress increments below the Mahembi building, one of the most tilted buildings in Santos, which settled and tilted significantly after the construction of the nearby Paineiras building. You will use the FEM software MIDAS GTS NX to assess the vertical stress increments produced by the Mahembi building alone (**Loading condition 1**, “small building only”), and by the two buildings

together (**Loading condition 2**, “small and large buildings”). Using Midas GTS NX, you will be able to visualize the stress increment bulbs due to the Mahembi building and superimpose the stresses produced by the construction of the Paineiras building. You should be able to relate the stress distribution in the ground with the tilting of the buildings.

The mesh used for the analysis is shown below. Note that this is a **2D plane strain model**, which means that the loading extends infinitely in the direction perpendicular to the paper, and that there is no out of plane (out of paper) deformation. The same loads (100 kPa for the small building, and 120 kPa for the large building) used in Lab #7 are used here for the analysis.



**Figure 3.** Mesh used for analysis of stress increments produced under loads due to buildings.

You will perform analyses for three different soil profiles that differ in the properties of the materials used for the different layers, specifically:

**Soil profile 1** is characterized by a uniform sand layer (which encompasses the entire mesh shown in Figure 3) with  $E_{\text{sand}}=50,000$  kPa.

In **Soil profile 2**, top 10 m of the same sand are underlain by soft clay with  $E_{\text{clay}}=2,500$  kPa. This is the scenario more representative of the actual soil conditions in Santos.

In **Soil profile 3**, top 10 m of the same sand are underlain by a very stiff rock layer with  $E_{\text{rock}}=5,000,000$  kPa.

## Report

No report is required for this lab. You are required to complete the questionnaire by the end of lab and hand it in.

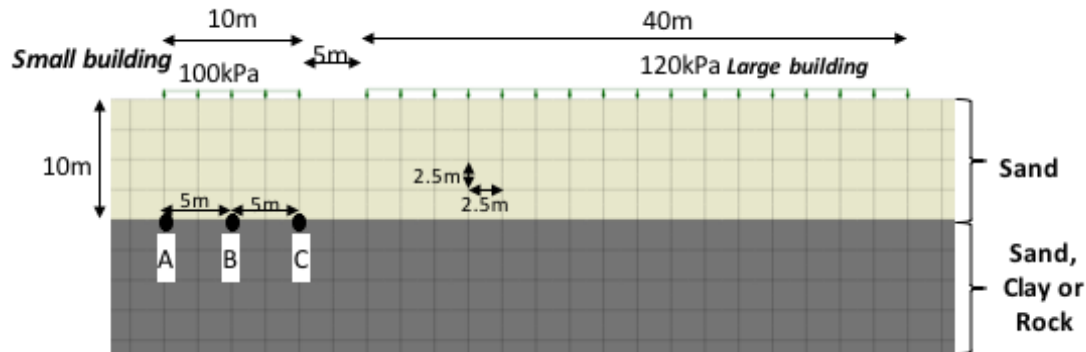
## Procedure:

### PART A – Analysis for uniform ground conditions (Soil profile 1): COMPARISON OF MIDAS RESULTS TO SOLUTION FROM LAB #7

The input file for this analysis is provided to you and you need to load it into Midas as described in Slide 2 of the tutorial. In this file, both layers have the same  $E=50,000$  kPa. Save the file with a new name (e.g. *StressAnalysisLab-SOILPROFILE 1.gts*).

- 1) Run the analysis as described in Slide 4 of the tutorial. The analysis provides you results for two cases: loading condition 1 (small building alone), and loading condition 2 (both buildings).
- 2) Visualize the stress bulbs for  $\Delta\sigma_v$  due to loading condition 1. Note that  $\Delta\sigma_v$  in Midas corresponds to the s-yy TOTAL component. See Slides 5 & 6 for how to do this (note that you can toggle between the stress bulb showing the effect of loading condition 1 and 2).  
The stress bulbs should look similar to those you were shown in class and in lab for different loading geometries. Zoom in on the bulbs close to the load. Using *StressAnalysis-Worksheet.pdf*, answer the following question:  
A1 - Why do you think the bulbs are not as “smooth” as those presented in class? How could the analysis be modified to make them smoother?
- 3) Refer to points A, B and C in Figure 4. Extract the values of  $\Delta\sigma_v$  at these points. See slide 7 in the tutorial for how to extract these values using the “probe” tool. Make sure you are getting the stresses at the correct nodes (in this part of the mesh the grid is 2.5 m x 2.5 m) and make sure the results you are extracting make physical sense. Do so for the case of loading by the small building alone (loading condition 1), and the case for loading by both buildings (loading condition 2). Copy by hand these values into *StressAnalysis-Worksheet.pdf* below the data obtained last week using the elastic solutions (Tables 1 and 2). Convert to positive values (see note below for reason).

**NOTE:** the software provides stresses using the mechanics sign convention (i.e. negative for compression). Convert these to positive when reporting them in your report, as in geotechnical engineering compressive stresses are positive.



**Figure 4.** Location of points A, B and C at which to derive stress increments.

- 4) Using the worksheet provided to you, answer the following questions:  
A2 - How do the results obtained using the elastic solution in Lab 7 compare with the numerical results?  
A3 - What are the basic assumptions of the solution that you used in Lab 7 to derive the stress increments at A, B and C?  
A4 - State two differences between the basic assumptions of the solution used in Lab 7 and those of the FEM used here for calculating  $\Delta\sigma_v$  (both are based on elastic behavior, so that would NOT be a difference). Why are the results for  $\Delta\sigma_v$  similar despite these differences?

## **PART B – Analysis for uniform ground conditions (Soil profile 1): EFFECT OF LARGE BUILDING**

- 1) Visualize the stress bulbs again (note that you can toggle between the stress bulb showing the effect of loading conditions 1 and 2). Using the sheet provided to you:
  - B1 – Determine how deep under the centerline of the small building the bulb corresponding to 50 kPa extends. Do this for loading conditions 1 and 2. Note that you may need to adjust the scale. What does the difference in the two depths imply?
- 2) Focusing on the results obtained using Midas compare the stress increases at A, B, and C due to loading conditions 1 and loading conditions 2. Using the sheet provided to you, answer the following questions:
  - B2 - By how much do the stresses at A, B and C increase as a result of the load associated with the large building? How do you explain the different increases at A, B and C?
  - B3 - How do the vertical stress increases at A, B and C from Table 2 explain the lean of the small building?
- 3) Extract the values of  $\Delta\sigma_v$  on a horizontal plane 10 m below the surface. See Slide 8 in the tutorial for how to proceed to obtain a stress diagram. Do so for both loading conditions 1 and 2. Copy the data in the first sheet (named “Dsv at 10 m depth”) of *StressAnalysis-Spreadsheet.xlsx* (beginning at row 6 in columns A-E and H-L, respectively). Save the file.
- 4) Visualize the variation of  $\Delta\sigma_v$  as a function of the position relative to the center of the building in the graph displayed in the sheet of the excel file named “Graph 1.” Identify the data points corresponding to A, B and C and check the numbers you inserted in Tables 1 and 2 for **Soil profile 1**. On the worksheet provided to you, and using Graph 1, answer the following questions:
  - B4 - Based on Graph 1, approximately, how far from the centerline of the small building does the effect of the large building stop being felt (state your criterion)?
  - B5- Using Graph 1, provide an estimate of  $\Delta\sigma_v$  below the left and right edges of the large building due to the loads from both buildings (no additional analysis needed, use results from Graph 1). Based on these values, would you expect the large building to exhibit similar, smaller or greater lean than that of the small one? Explain.
- 5) Extract the vertical stress increments under the centerline of the Mahembi (small) building. See Slide 7 in the tutorial for how to proceed. Do this for both loading conditions 1 and 2. Copy values in the second sheet of *StressAnalysis-Spreadsheet.xlsx* (sheet named “Dsv under CL of small”) beginning at row 6 for columns A-E (loading condition 1) and H-L (loading condition 2).
- 6) Visualize the variation of  $\Delta\sigma_v$  as a function of depth under the centerline of the small building in the sheet of the excel file named “Graph 4.” Using *StressAnalysis-Worksheet.pdf*, answer the following:
  - B6 - Based on Graph 4, how deep under the centerline of the small building does  $\Delta\sigma_v$  exceed 50 kPa? Do this for loading condition 1 and loading condition 2. Your answer should be consistent with that provided for B1.
- 7) Using the sheet provided to you, answer the following:

**Note: you will not be evaluated based on whether your answers to B7 and B8 are correct. Please do not make corrections.**

B7 - If you performed a new analysis replacing sand with soft clay over the entire depth of the model, how would you expect the stresses at A, B and C to change? Why?

B8 – How would you expect the resulting deformations in the soil mass to change? Why?

### **PART C – Analysis of the effects of layers of different stiffness: Soil profile 2 (soft clay with $E = 2500$ kPa below sand)**

- 1) In Soil profile 2, the stratigraphy is modified to include a layer of soft clay below 10 m. Before conducting any analysis, using *StressAnalysis-Worksheet.pdf*, answer the following question:

C1- For loading condition 1, do you predict that the values of  $\Delta\sigma_v$  at A, B and C will increase, be the same or decrease compared to the values determined at the same points for soil profile 1. What is the rationale?

**Note: you will not be evaluated based on whether your answers for C1 are correct. Please do not make corrections, if you later find your answer for C1 incorrect.**

- 2) Conduct analysis for Soil profile 2. To start, open the original file and save it using a different name (e.g. *StressAnalysisLab-SOILPROFILE 2.gts*). You will need to edit the input file and modify the elastic modulus of the bottom layer. See Slide 9 in the tutorial for details on how to do this.
- 3) Run the analysis. Again you will obtain results for the stress increments produced by loading conditions 1 and 2.
- 4) Visualize the stress bulb under the small building for the condition corresponding to loading condition 1. Using *StressAnalysis-Worksheet.pdf*, answer the following question:

C2 – Describe what differences you can identify in the pressure bulbs between Soil profile 1 and Soil profile 2? Make sure you use the same scale for both.

- 5) Extract the values of  $\Delta\sigma_v$  on a horizontal plane 10 m below the surface. As above, refer to slide 8 of the tutorial and copy the data in the first Sheet (with name “Dsv at 10 m depth”) of *StressAnalysis-Spreadsheet.xlsx* (columns P-T for small building alone and W-AA for two buildings). Save the file.
- 6) Visualize the variation of  $\Delta\sigma_v$  as a function of the position relative to the center of the building in “Graph 2” of the excel file. Using *StressAnalysis-Worksheet.pdf*, answer the following questions:

C3 - Relative to Soil profile 1, what is the effect of the presence of a soft clay layer under the sand on the variation of  $\Delta\sigma_v$  as a function of distance from the centerline of the small building (Graph 2)?

C4 - In Graph 2, consider the areas between the x axis and the blue (Soil profile 1) curve and between the x axis and the orange curve (Soil profile 2). What has to be true about these areas? Think in terms of equilibrium.

- 7) Extract values of  $\Delta\sigma_v$  at points A, B and C using the “probe” tool (see slide 7 of tutorial). Copy these values in Tables 1 and 2 on the sheet provided to you. Using *StressAnalysis-Worksheet.pdf*, answer the following questions:

C5 - For loading condition 1 compare the values of  $\Delta\sigma_v$  at A, B and C to those  $\Delta\sigma_v$  obtained for Soil profile 1. What do you observe?

C6 – Would you use the elastic solutions employed in Lab 7 to predict stress increases in a soil profile similar to that of Soil profile 2? Would this provide an accurate estimate?

Note that the effect of the presence of a soft layer under the top 10 m of sand is evident also when one examines the values of  $\Delta\sigma_v$  produced by both buildings (e.g. you can check Graph 3 or compare the pressure bulbs).

- 8) Extract the vertical stress increments under the centerline of the Mahembi (small) building. See Slide 8 in the tutorial for how to proceed. Do this for both loading conditions. Copy values in the second sheet of *StressAnalysis-Spreadsheet.xlsx* (sheet named “Dsv under CL of small”) in columns P-T (small building only) and W-AA (both buildings).
- 9) Visualize the variation of  $\Delta\sigma_v$  as a function of depth under the centerline of the small building in the sheet of *StressAnalysis-Spreadsheet.xlsx* named “Graph 4.” Using *StressAnalysis-Worksheet.pdf*, answer the following:

C7 - For Soil profile 2, how deep under the centerline of the small building does  $\Delta\sigma_v$  exceed 50 kPa? Do this for both loading conditions. How do these values compare to those for Soil profile 1 that you provided for B1?

- 10) Refer to the data you summarized in Tables 1 and 2 in *StressAnalysis-Worksheet.pdf*.

C8 - Refer to the data you summarized in Tables 1 and 2. At point B calculate the ratio of  $\Delta\sigma_v$  due to loading condition 2 (from Table 2) to  $\Delta\sigma_v$  due to loading condition 1 (from Table 1). Calculate the ratio for Soil profile 1 and Soil profile 2. What do you observe? For which soil conditions is the effect of the second building more significant?

## **PART D – Analysis of the effects of layers of different stiffness: Soil profile 3 (stiff rock with $E = 5 \times 10^6$ kPa below sand) – Focus exclusively on loading condition 1**

- 1) In Soil profile 3, the stratigraphy is modified to include a stiff rock layer below 10 m. Before conducting any analysis, using *StressAnalysis-Worksheet.pdf*, answer the following question:

**Note: you will not be evaluated based on whether your answers for D1 are correct. Please do not make corrections, if you later find your answer for D1 incorrect.**

D1- For loading condition 1, do you predict that the values of  $\Delta\sigma_v$  at A, B and C will increase, be the same or decrease compared to the values determined at the same points for Soil profile 1. What is your rationale?

- 2) Conduct the analysis for Soil profile 3. To start, open the original file and save it using a different name (e.g. *StressAnalysisLab-SOILPROFILE 3.gts*). You will need to edit the input file and modify the elastic modulus of the bottom layer. See Slide 9 in the tutorial for details on how to do this.
- 3) Extract the values of  $\Delta\sigma_v$  at points A, B and C using the “probe” tool (see slide 7 of tutorial), and copy them in Table 1 (loading condition 1) and Table 2 (loading condition 2) on *StressAnalysis-Worksheet.pdf*. Using *StressAnalysis-Worksheet.pdf* answer the following question:

D2 - For loading condition 1 compare the values of  $\Delta\sigma_v$  at A, B and C to those  $\Delta\sigma_v$  on obtained for Soil profile 1. What do you observe?

- 4) Extract the values of  $\Delta\sigma_v$  on a horizontal plane 10 m below the surface. For this part focus on the results for loading condition 1. As above, refer to slide 8 of the tutorial and copy the data in the first Sheet (with name “Dsv at 10 m depth”) of *StressAnalysis-Spreadsheet.xlsx* (columns AD-AH). Save the file.
- 5) Visualize the variation of  $\Delta\sigma_v$  as a function of the position relative to the center of the building in “Graph 2” of the excel file. Using *StressAnalysis-Worksheet.pdf* answer the following question:



- D3 - Relative to soil profile 1, what is the effect of the presence of a stiff layer below the sand on the variation of  $\Delta\sigma_v$  as a function of distance from the centerline of the small building (Graph 2)?

## **PART E – Design implications (based on Soil profile 2 results)**

Using *StressAnalysis-Worksheet.pdf* answer the following questions that illustrate the practical significance and the design implications of the analyses and calculations you performed.

- E1 - Refer to the results for Soil profile 2. Assume that the clay layer can support a vertical stress increment of **50kPa** without undergoing large deformations. Focusing on what happens under the centerline of the small building, would you expect large settlements in the clay, under the load due the small building alone? Why?
- E2 - Focusing on what happens under the centerline of the small building, would you expect large settlements in the clay, under the load due to both buildings? Why?
- E3 - Imagine you are a Geotechnical Engineer in Santos in the early 50's, when the construction of tall buildings started, and no technology for the construction of deep foundations applicable to the local site conditions is available. Assume, as above, that the clay layer can support a total  $\Delta\sigma_v$  of no more that **50kPa** without undergoing large deformations, and that each story of a building produces a stress of 10kN/m<sup>2</sup>. Provide an estimate of the number of stories that you would recommend for the Paineiras (large) building to avoid large settlements underneath the centerline of the Mahembi (small) building.

You should not perform additional analyses with Midas. Base your answer on your knowledge of the  $\Delta\sigma_v$  associated with the small building (this value does not change), and take advantage of the fact that the solution provided by Midas is based on elasticity, i.e. stress increments are proportional to the loads applied at the surface. Show your work.



# Laboratory on Stress Analysis using FEM

Tutorial for conducting the analysis using the Midas GTS NX software

Prepared by O. Vitali, A. Getchell and M. Santagata – Spring 2020

*to be used in conjunction with Lab Master Handout*

# Step 1: GETTING STARTED

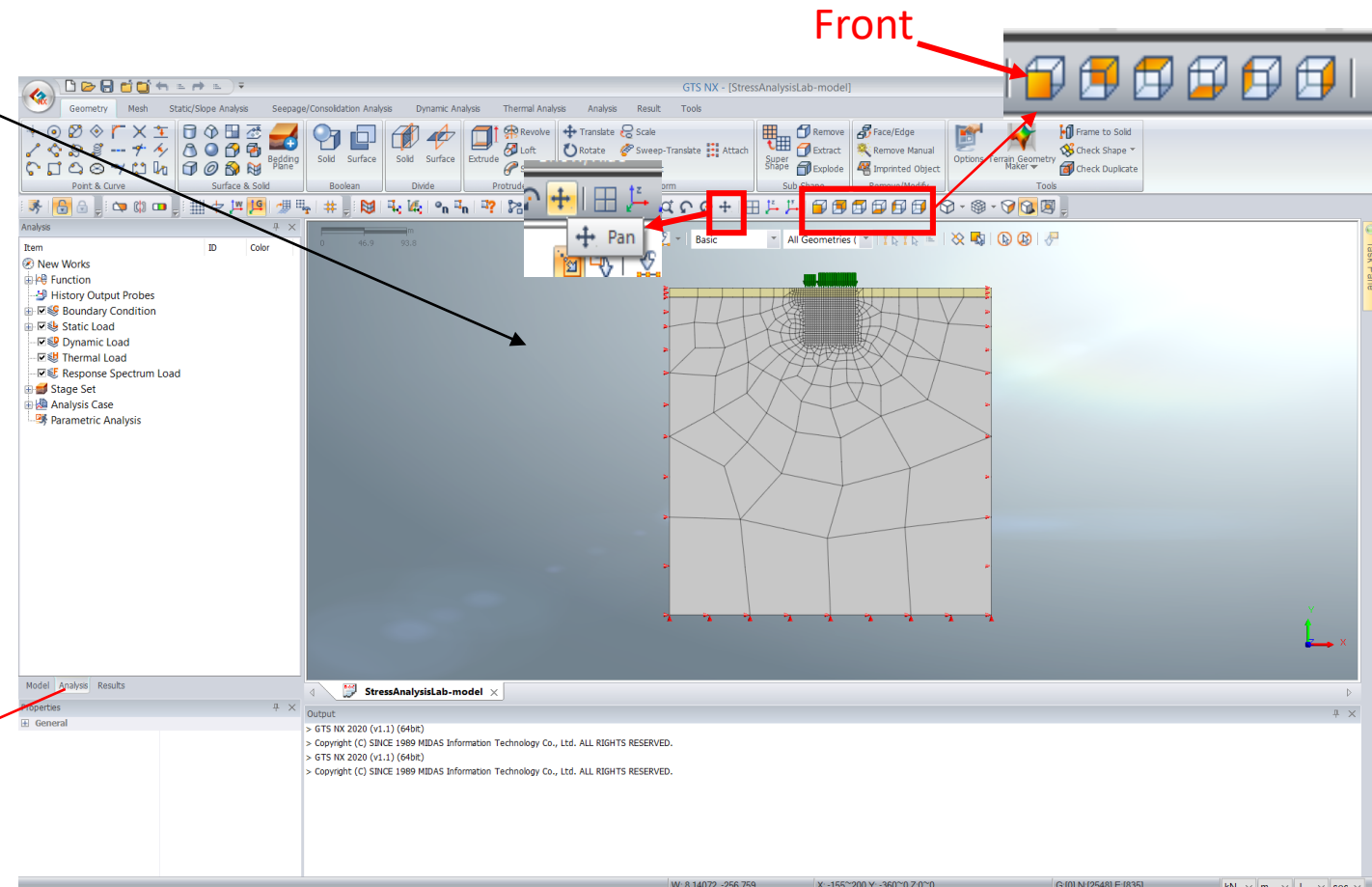
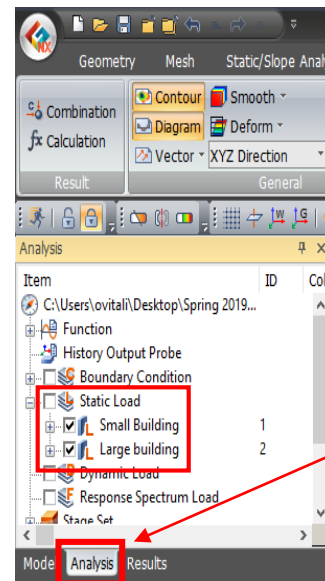
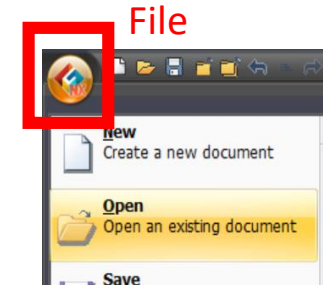
## 1a. Loading the model

Open StressAnalysisLab-model.gts (double click on the file or go to File → Open and select the file)  
This is how the model will look.

You may use the cursor to zoom in and out, and the Pan tool to drag the window.  
Hint: Click on “Front” to center the model.

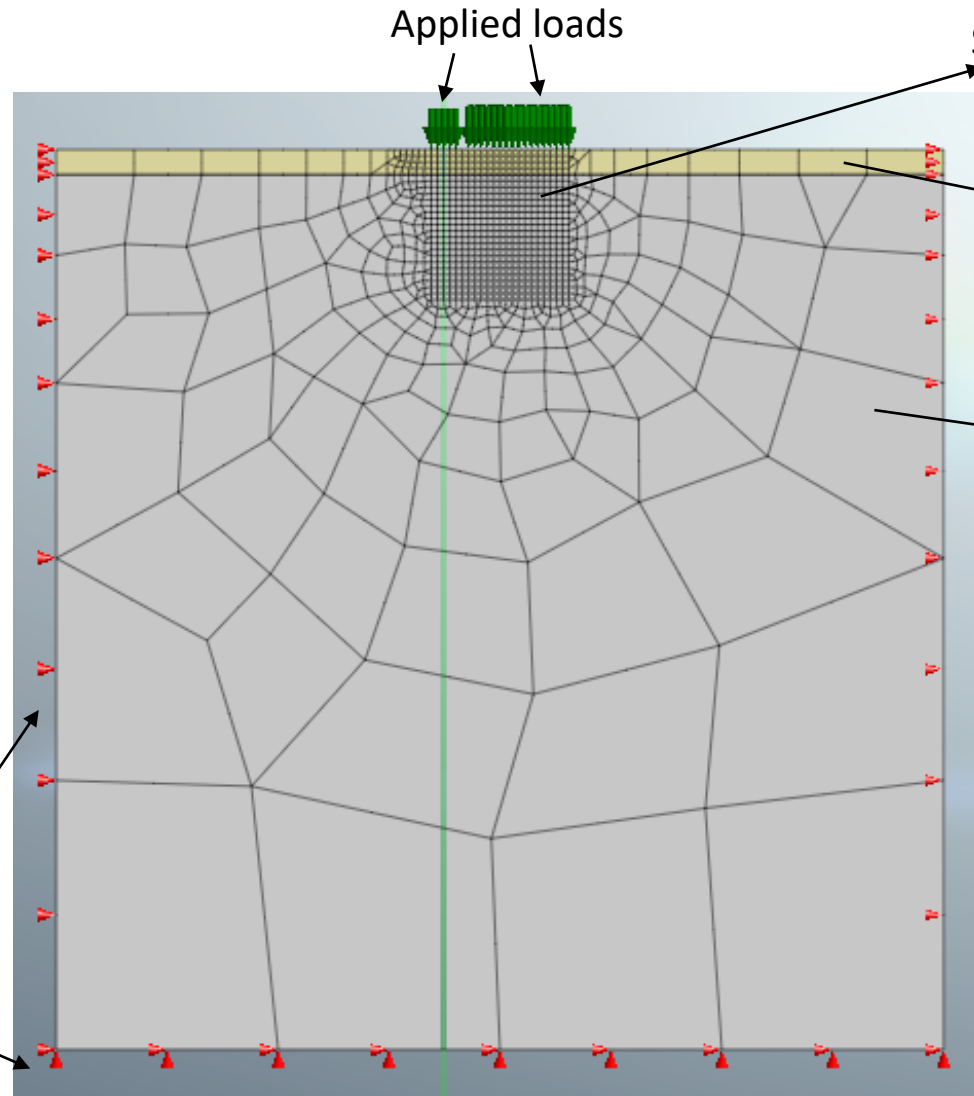
If you do not see the load applied at the surface, check the boxes “Small Building” and “Large Building” by going to:

Analysis → Static Load → Small Building and Large Building



# Step 1: GETTING STARTED

## 1b. Inspecting the model: FEM mesh, boundary conditions, and materials



Applied loads

Size of elements = 2.5 m x 2.5 m

Top layer – Linear elastic material  
 $E=50\text{MPa}$ ,  $\nu=0.3$

Bottom layer – Linear elastic material  
 $E=\text{variable}$ ,  $\nu=0.3$

Note variable size of elements

- Each analysis will generate two sets of results:
- One for the small building alone (Loading condition 1)
  - One for both buildings (Loading condition 2)

Displacement constraints

## Step 2: PERFORMING THE ANALYSIS

To run the model, go to “Analysis” and click on “Perform” and “Ok”.

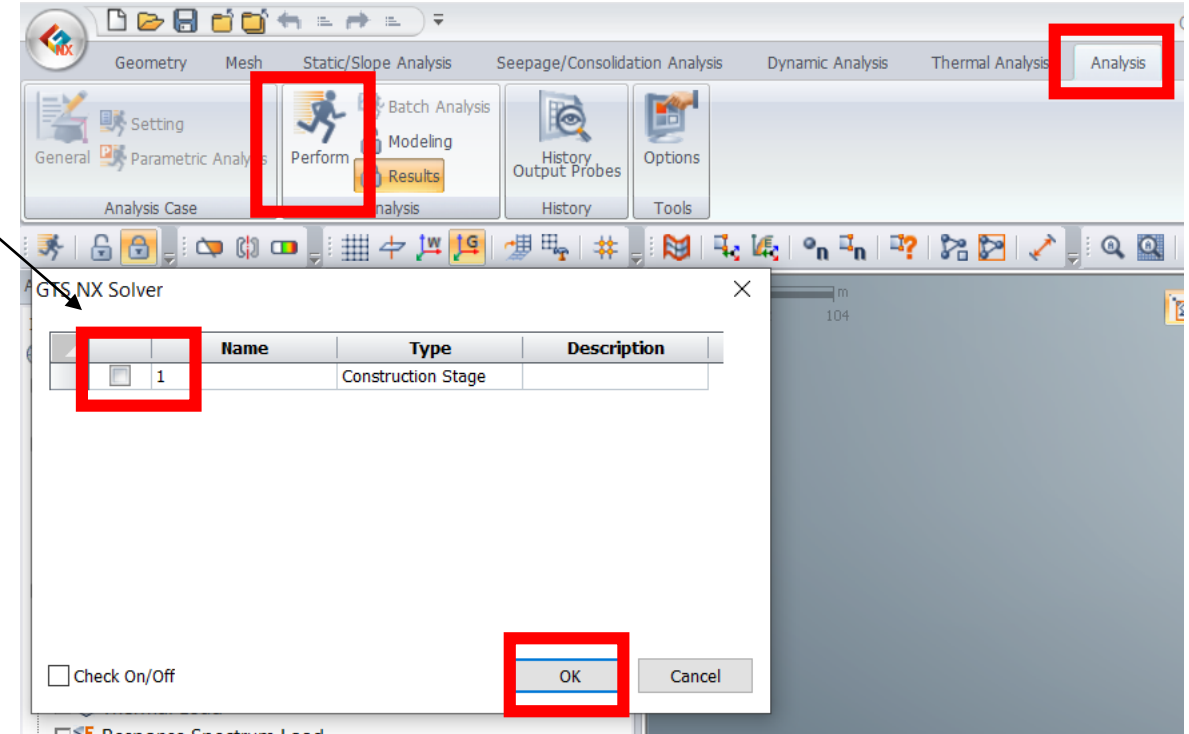
Make sure this box is checked.

The program will calculate two solutions:

- One for the small building alone (Loading condition 1)
- One for both buildings (Loading condition 2)

### IMPORTANT NOTES:

- The results are presented using the mechanics sign convention, i.e. compressive stresses are negative. When reporting them, you will need to change their sign.
- All stresses reported are **STRESS INCREMENTS**, i.e. the stresses produced by the applied loads (these would be added to the initial stresses).



# Step 3: ANALYZING THE RESULTS

## 3a. Visualizing the contours of vertical stress increments

**NOTE: FOR EACH SOIL PROFILE YOU NEED TO EXTRACT THE STRESSES FOR THE TWO CASES OF INTEREST - SMALL BUILDING ONLY AND BOTH BUILDINGS**

The model calculates the vertical stress increments for the case in which there is only the Mahembi building (“small building only” or Loading condition 1) and in presence of both buildings (Loading condition 2).

For each case, to see the distribution of the vertical stress increments, go to: RESULTS (see in red frame below) → “Small building only” or “Small and Large building” → Plane strain stresses → S-YY (this is the vertical stress increment).

**Small Building only**

- INCR=1 (LOAD=1.000)
- Displacements
- Grid Forces
- Reactions
- Plane Strain Forces
- Plane Strain Stresses
  - S-XX TOTAL
  - S-YY TOTAL
  - S-ZZ TOTAL
  - S-XX
  - S-YY
  - S-ZZ

**Small and Large buildings**

- INCR=1 (LOAD=1.000)
- Displacements
- Grid Forces
- Reactions
- Plane Strain Forces
- Plane Strain Stresses
  - S-XX TOTAL
  - S-YY TOTAL
  - S-ZZ TOTAL
  - S-XX
  - S-YY
  - S-ZZ
  - S-XY
  - S-MAJOR PRINCIPAL (V)
  - S-MINOR PRINCIPAL (V)
  - SAFETY FACTOR
  - S-MAX SHEAR
  - S-VON MISES
  - MEAN EFFECTIVE PRESSU..
  - MEAN TOTAL PRESSURE
  - PORE STRESS
  - EXCESSIVE PORE STRESS

**Results**

*Zoom in sufficiently to see what is happening close to the surface*

*You can toggle between the two plots using this slider*

*Note that two figures may not use same scale*

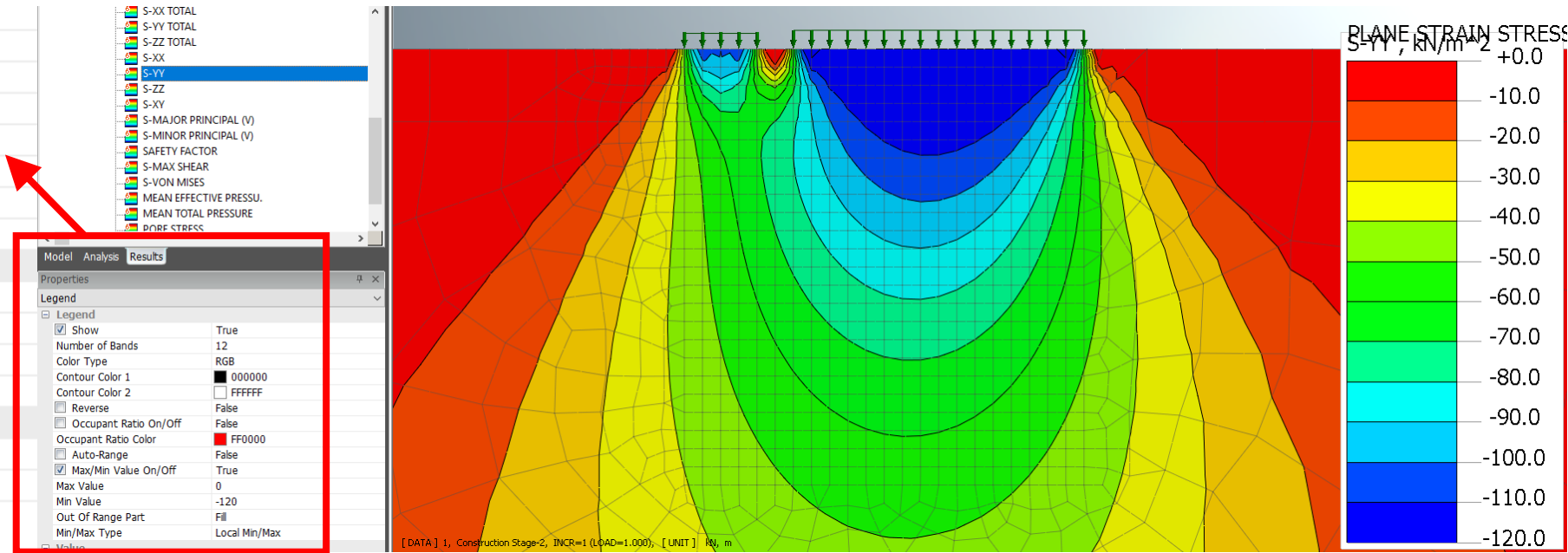
# Step 3: ANALYZING THE RESULTS

## 3a. (cont.) Visualizing the contours of vertical stress increments

The screenshot shows the 'Properties' dialog box for the 'Results' tab. The 'Legend' section is expanded, and the 'Max/Min Value On/Off' checkbox is checked. The 'Number of Bands' is set to 12. The 'Max Value' is set to 0 and the 'Min Value' is set to -120. The 'Value' section shows 'Value Color' as black (000000) and 'Value Exponential' as False. The 'Background' section shows 'Background Color' as white (FFFFFF).

Property	Value
Legend	Expanded
Show	True
Number of Bands	12
Color Type	RGB
Contour Color 1	000000
Contour Color 2	FFFFFF
Reverse	False
Occupant Ratio On/Off	False
Occupant Ratio Color	FF0000
Auto-Range	False
Max/Min Value On/Off	True
Max Value	0
Min Value	-120
Out Of Range Part	Fill
Min/Max Type	Local Min/Max
Value	Expanded
Value Color	000000
Value Exponential	False
Value Decimal Point	1
Value Font Size	25
Background	Expanded
Background	Solid Fill
Color	FFFFFF

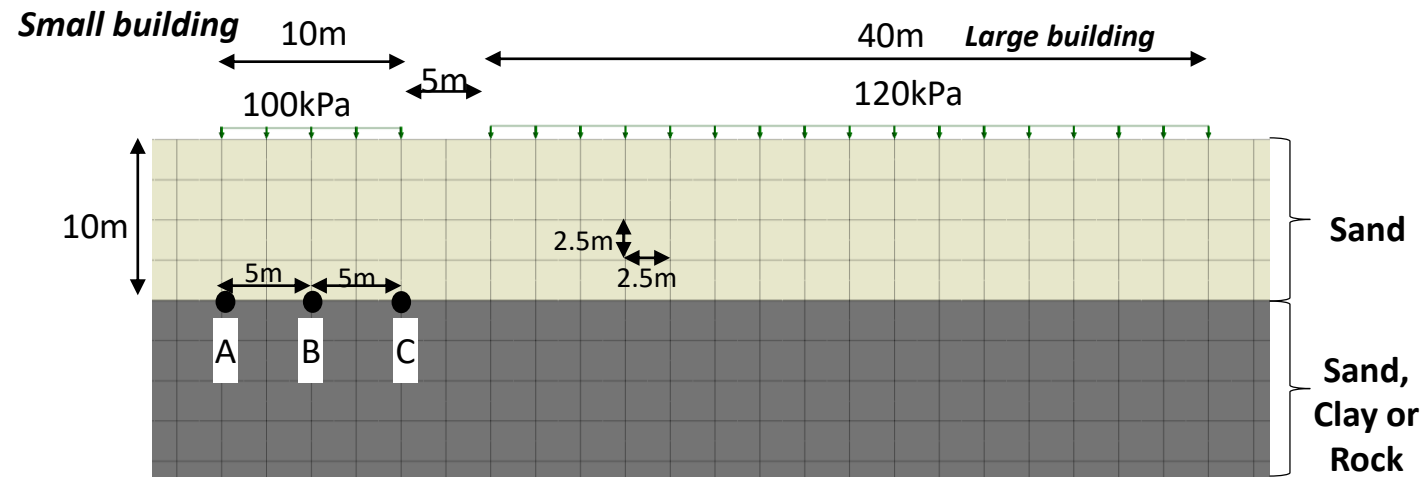
To better visualize the stress bulbs, adjust the scale and the number of bands. To adjust the scale, go to: Properties → Legend: check Max/Min Value and enter Max Value and Min Value.



# Step 3: ANALYZING THE RESULTS

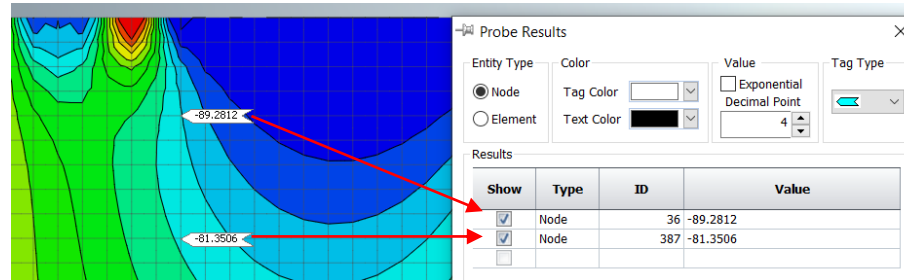
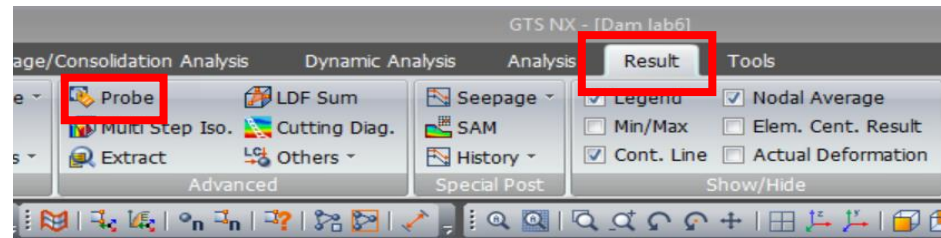
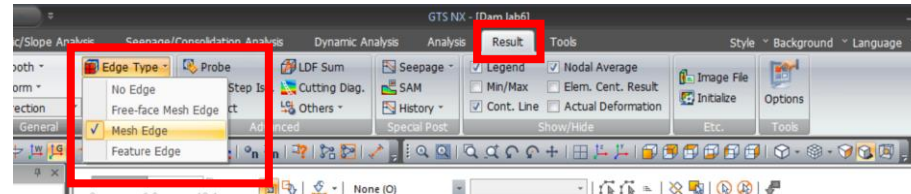
## 3b. Extracting stresses at specific points

For all soil profiles examined, you are asked to extract the vertical stress increments at nodes A, B and C (see location on the right).

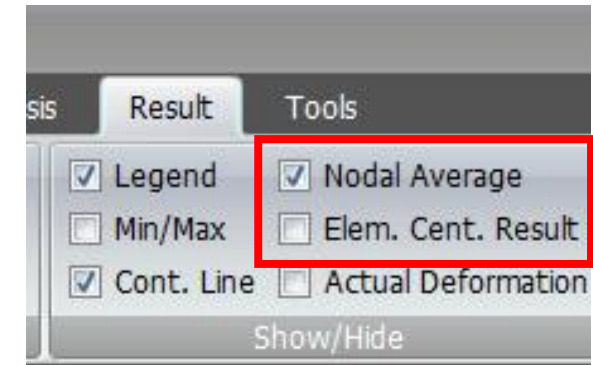


To get the stresses at the nodes, use the **probe tool**:

- 1) Make sure the grid is visible (Results – Edge Type - Mesh Edge)
- 2) Results → Probe
- 3) Select the nodes and view values in table
- 4) Copy values of stress increments for each scenario to excel sheet (remember to convert the stress increments to positive)



Make sure that the box “Nodal Average” is checked and the box “Elem. Cent. Result” is unchecked.





# Step 3: ANALYZING THE RESULTS

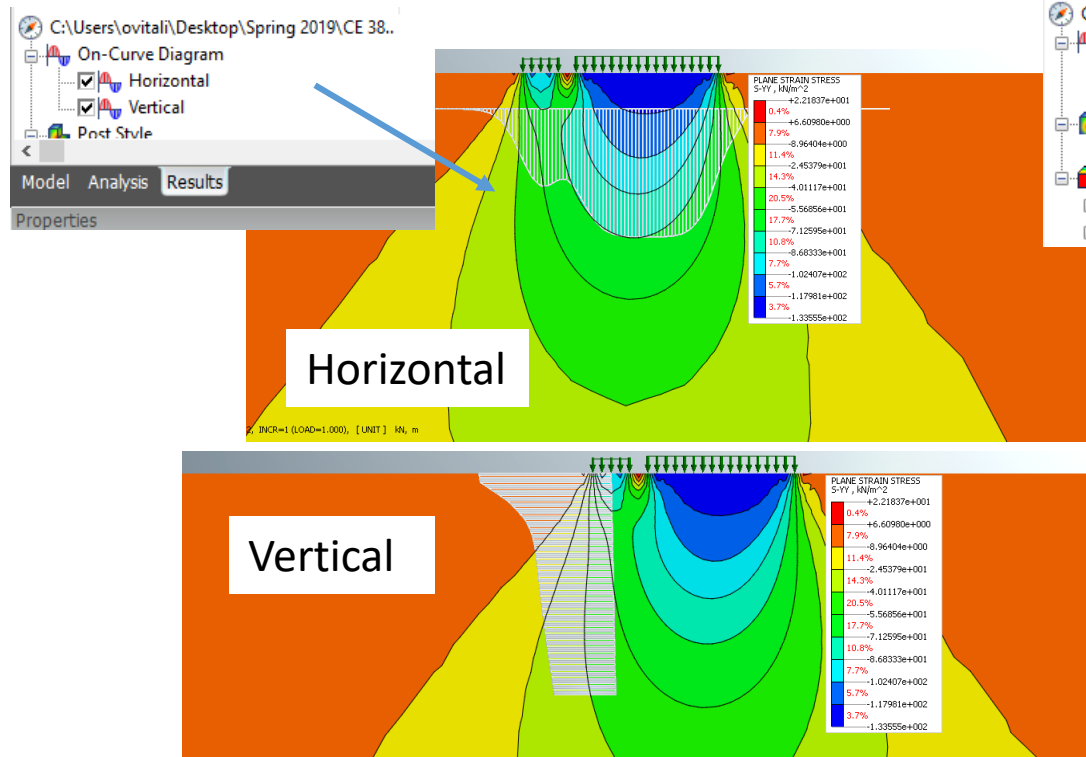
## 3c. Obtaining stress diagrams

For two of the soil profiles modeled you will need to get the horizontal stress diagram at the top of the bottom layer and the vertical stress diagram below the center of the small (Mahembi) building.

The horizontal and vertical cross-sections to extract the stresses have been already built.

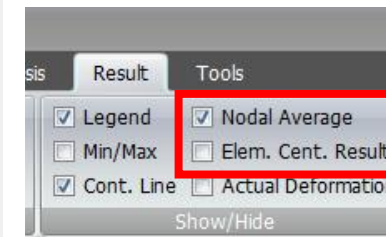
To extract the results in table format: go to Results → On-Curve Diagram (check the boxes to see the diagrams) → click with right button on “Horizontal” or “Vertical” → Show Table → Copy the values to *StressAnalysis-Spreadsheet.xlsx* → Close table window

**NOTE: FOR EACH SOIL PROFILE EXTRACT THE STRESS DIAGRAMS FOR THE TWO LOADING CONDITIONS: SMALL BUILDING ONLY AND BOTH BUILDINGS**



ID	X (m)	Y (m)	Z (m)	Value
1	-20.7721	-10	0	-1.891032e+001
2	-49.7599	-10	0	-1.842919e+001
3	-48.7478	-10	0	-1.994495e+001
4	-47.7356	-10	0	-2.146891e+001
5	-46.7235	-10	0	-2.299277e+001
6	-45.7113	-10	0	-2.451663e+001
7	-44.6992	-10	0	-2.603349e+001
8	-43.687	-10	0	-2.823445e+001
9	-42.6749	-10	0	-3.099990e+001
10	-41.8627	-10	0	-3.376536e+001
11	-40.6506	-10	0	-3.653081e+001
12	-39.6384	-10	0	-3.929627e+001
13	-38.6263	-10	0	-4.206172e+001
14	-37.6141	-10	0	-4.482718e+001
15	-36.602	-10	0	-4.759264e+001
16	-35.5899	-10	0	-5.03581e+001
17	-34.5777	-10	0	-5.312356e+001
18	-33.5655	-10	0	-5.588902e+001
19	-32.5534	-10	0	-5.865448e+001
20	-31.5412	-10	0	-6.141994e+001
21	-30.5291	-10	0	-6.41854e+001
22	-29.5169	-10	0	-6.695086e+001
23	-28.5048	-10	0	-6.971632e+001
24	-27.4926	-10	0	-7.248178e+001
25	-26.4805	-10	0	-7.524724e+001
26	-25.4683	-10	0	-7.80127e+001
27	-24.4562	-10	0	-8.077816e+001
28	-23.444	-10	0	-8.354362e+001
29	-22.4319	-10	0	-8.630908e+001
30	-21.4197	-10	0	-8.907454e+001
31	-20.4076	-10	0	-9.183999e+001
32	-19.3954	-10	0	-9.460545e+001
33	-18.3833	-10	0	-9.737091e+001
34	-17.3711	-10	0	-1.00126e+002
35	-16.359	-10	0	-1.025429e+002
36	-15.3468	-10	0	-1.049598e+002

Make sure that the box “Nodal Average” is checked and the box “Elem. Cent. Result” is unchecked.



# Step 4: MODIFYING THE MODEL

To change the material properties for one of the layers (required to perform analysis for Part C and D)

1) Click Pre-Mode (opened locker);

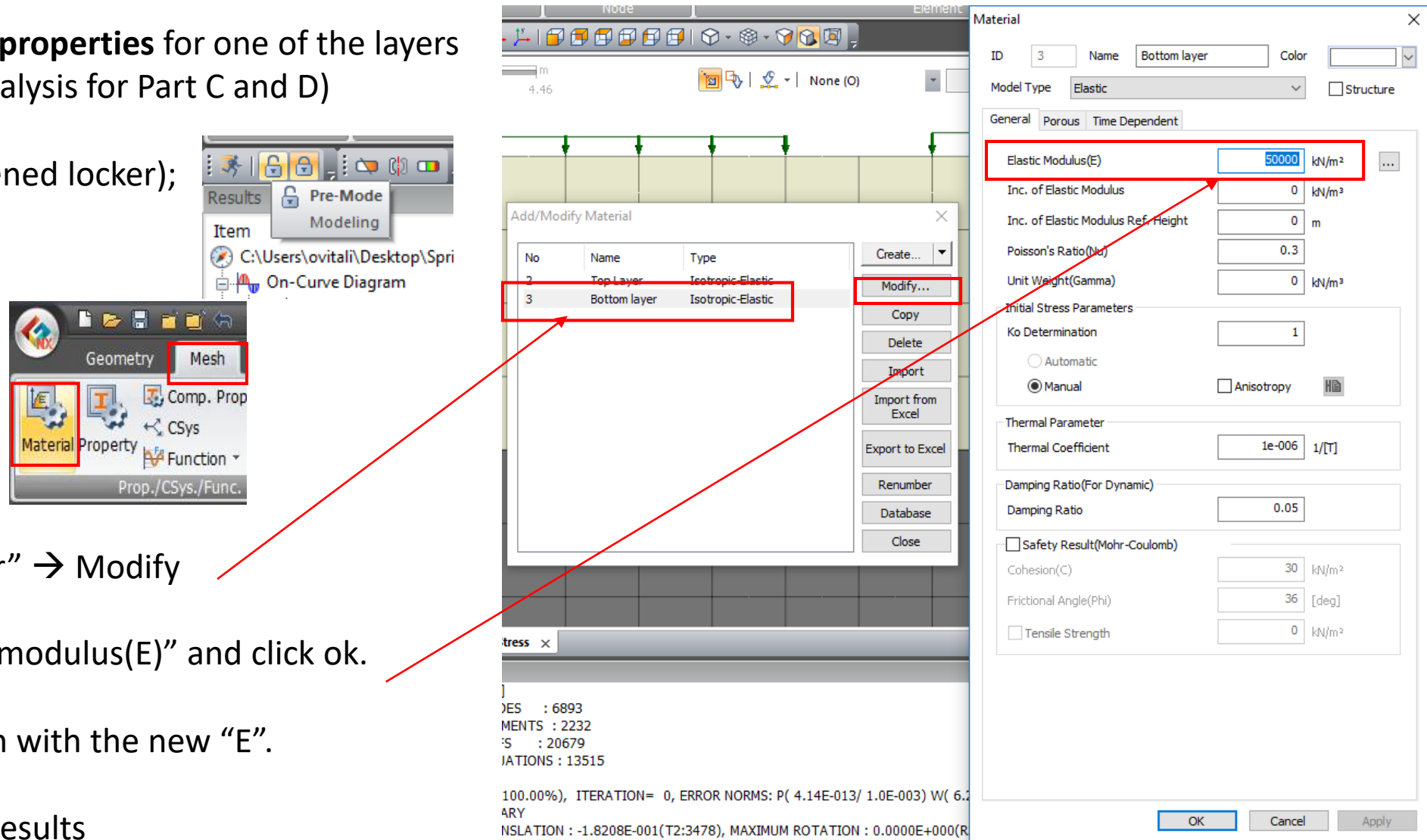
2) Mesh → Material

3) Select “Bottom layer” → Modify

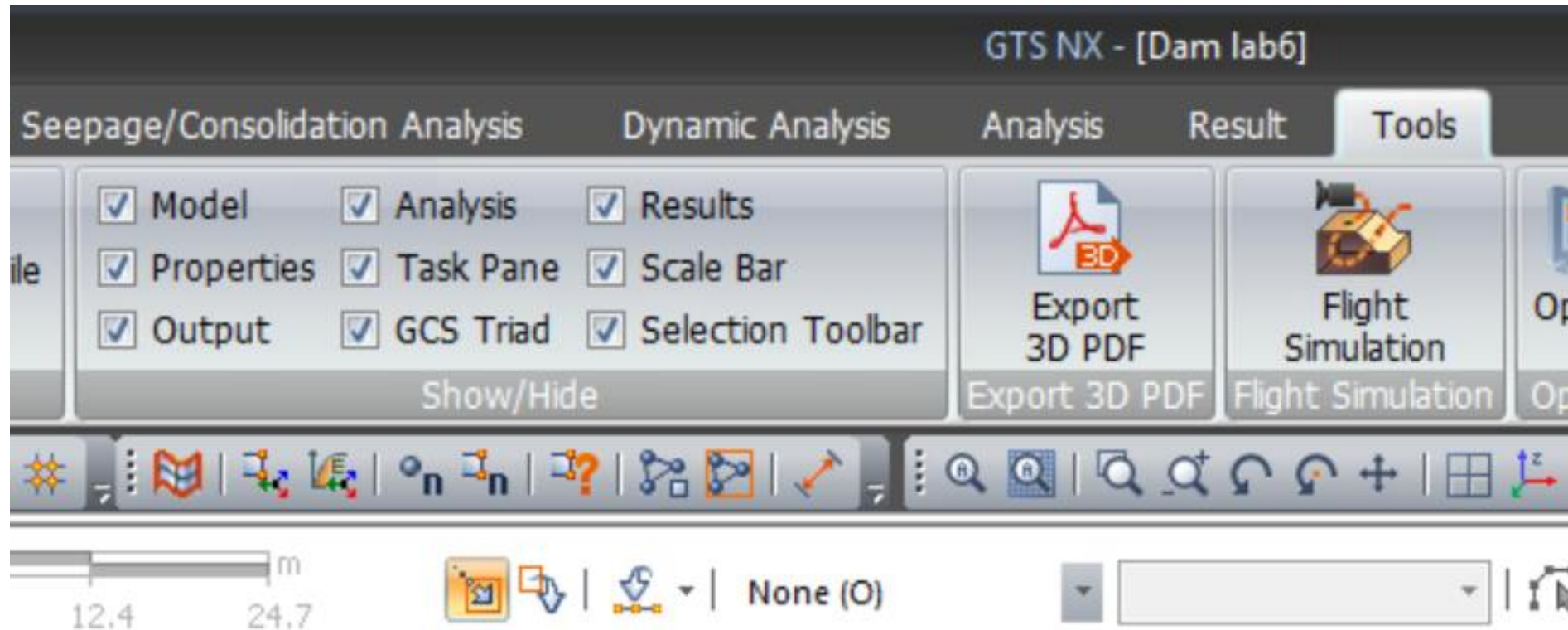
4) Change the “Elastic modulus(E)” and click ok.

5) Run the model again with the new “E”.

6) Analyze new set of results



FINAL HINT: If you close one of the “working windows” (probably by mistake), you can reopen it in Tools.



**Laboratory on Stress Analysis using FEM  
WORKSHEET TO BE TURNED IN AT THE END OF LAB**

**SUMMARY TABLES:**

**Table 1. Vertical stress increases - Contribution of small building only (Loading condition 1)**

	POINT A (kPa)	POINT B (kPa)	POINT C (kPa)
<i>Elastic Solution (Lab 7)</i>	<b>40.4</b>	<b>54.5</b>	<b>40.4</b>
<i>Midas - Soil Profile 1</i>			
<i>Midas - Soil Profile 2</i>			
<i>Midas - Soil Profile 3</i>			

**Table 2. Vertical stress increases - Contribution of small and large buildings (Loading condition 2)**

	POINT A (kPa)	POINT B (kPa)	POINT C (kPa)
<i>Elastic Solution (Lab 7)</i>	<b>44.6</b>	<b>64.4</b>	<b>66.2</b>
<i>Midas - Soil Profile 1</i>			
<i>Midas - Soil Profile 2</i>			
<i>Midas - Soil Profile 3</i>			

**PART A: Analysis for uniform ground conditions (Soil profile 1): COMPARISON OF MIDAS RESULTS TO SOLUTION FROM LAB 7**

A1) Why do you think the bulbs close to the surface are not as “smooth” as those presented in class? How could the analysis be modified to make them smoother?

.....

.....

A2) How do the results obtained using the elastic solution in Lab 7 compare with the numerical results?

.....

.....

A3) What are the basic assumptions of the solution that you used in Lab 7 to derive the stress increments at A, B and C?

.....  
.....

A4) State two differences between the basic assumptions of the solution used in Lab 7 and those of the FEM used here for calculating  $\Delta\sigma_v$  (both are based on elastic behavior, so that would NOT be a difference). Why are the results for  $\Delta\sigma_v$  similar despite these differences?

.....  
.....

**PART B: Analysis for uniform ground conditions (Soil profile 1): EFFECT OF LARGE BUILDING**

B1) Determine how deep under the centerline of the small building the bulb corresponding to 50 kPa extends? Do this for loading conditions 1 and 2.

For loading condition 1 ..... For loading condition 2 .....

What does the difference in the two depths imply? .....

.....

B2) By how much do the stresses at A, B and C increase as a result of the load applied by the large building (difference between values in Table 2 and Table 1) ?

A ..... kPa      B ..... kPa      C ..... kPa

How do you explain the differences in these values between A, B and C? .....

.....

B3) How do the vertical stress increases at A, B and C from Table 2 explain the lean of the small building?.....

.....

B4) Based on Graph 1, approximately, how far from the centerline of the small building does the effect of the large building stop being felt (state your criterion)?.....

.....

B5) Using Graph 1, provide an estimate of  $\Delta\sigma_v$  below the left and right edges of the large building due to the loads from both buildings (no additional analysis needed, use results from Graph 1):

Left edge ..... kPa      Right edge .....kPa

Based on these values, would you expect the large building to exhibit similar, smaller or greater lean than that of the small one? Explain

.....

.....

B6) Based on Graph 4, how deep under the centerline of the small building does  $\Delta\sigma_v$  exceed 50 kPa? Do this for loading condition 1 and loading condition 2.

For loading condition 1..... For loading condition 2.....

Check that your answer is consistent with your answer for B1?.....

B7) If you performed a new analysis replacing sand with a soft clay over the entire depth of the model, how would you expect the stresses at A, B and C to change? Why?

**Note: you will not be evaluated based on whether your answers to B7 and B8 are correct. Please do not make corrections.**

.....  
.....

B8) How would you expect the resulting deformations in the soil mass to change? Why?

.....  
.....

**PART C: Analysis of the effects of layers of different stiffness: Soil profile 2 (SOFT CLAY BELOW SAND)**

C1) For loading condition 1, do you predict that the values of  $\Delta\sigma_v$  at A, B and C will (circle your answer):

*increase*                      *be the same*                      *decrease*

compared to the values determined at the same points for soil profile 1. What is your rationale?

.....

**Note: you will not be evaluated based on whether your answers to C1 are correct. Please do not make corrections, if you later find your answer to C1 to be incorrect.**

C2) Describe what differences you can identify in the pressure bulbs between Soil profile 1 and Soil profile 2? Make sure you use the same scale for both.

.....  
.....

C3) Relative to Soil profile 1, what is the effect of the presence of a soft clay layer under the sand on the variation of  $\Delta\sigma_v$  as a function of distance from the centerline of the small building (Graph 2)?

.....  
.....

C4) In Graph 2, consider the areas between the x axis and the blue (Soil profile 1) curve and between the x axis and the orange curve (Soil profile 2). What has to be true about these areas? Think in terms of equilibrium. ....

.....

C5) For loading condition 1 compare the values of  $\Delta\sigma_v$  at A, B and C to those  $\Delta\sigma_v$  obtained for Soil profile 1. What do you observe? .....

C6) Would you use the elastic solutions employed in Lab 7 to predict stress increases in a soil profile similar to that of Soil profile 2? Would this provide an accurate estimate? .....

C7) For Soil profile 2, based on Graph 4, how deep under the centerline of the small building does  $\Delta\sigma_v$  exceed 50 kPa?

For loading condition 1 ..... For loading condition 2 .....

How do these values compare to those for Soil profile 1 that you provided for B1? .....

C8) Refer to the data you summarized in Tables 1 and 2. At point B calculate the ratio of  $\Delta\sigma_v$  due to loading condition 2 (from Table 2) to  $\Delta\sigma_v$  due to loading condition 1 (from Table 1). Calculate the ratio for Soil profile 1 and Soil profile 2.

Ratio for Soil profile 1 ..... Ratio for Soil profile 2 .....

What do you observe? For which soil conditions is the effect of the second building more significant? .....

**PART D: Analysis of the effects of layers of different stiffness: Soil profile 3 (STIFF ROCK BELOW SAND) – focus exclusively on loading condition 1**

D1) For loading condition 1, do you predict that the values of  $\Delta\sigma_v$  at A, B and C will:

*increase*                      *be the same*                      *decrease*

compared to the values determined at the same points for Soil profile 1. What is your rationale? .....

**Note: you will not be evaluated based on whether your answers to D1 are correct. Please do not make corrections, if you later find your answer to D1 to be incorrect.**

D2) For loading condition 1 compare the values of  $\Delta\sigma_v$  at A, B and C to those  $\Delta\sigma_v$  on obtained for Soil profile 1. What do you observe? .....

D3) Relative to soil profile 1, what is the effect of the presence of a stiff layer below the sand on the variation of  $\Delta\sigma_v$  as a function of distance from the centerline of the small building (Graph 2)?

.....  
.....

**PART E: DESIGN IMPLICATIONS (based on Soil profile 2 results)**

E1) Refer to the results for Soil profile 2. Assume that the clay layer can support a vertical stress increment of **50kPa** without undergoing large deformations. Focusing on what happens under the centerline of the small building, would you expect large settlements in the clay, under the load due the small building alone? Why?.....

.....

E2) Focusing on what happens under the centerline of the small building, would you expect large settlements in the clay, under the load due to both buildings? Why?

.....

.....

E3) Imagine you are a Geotechnical Engineer in Santos in the early 50's, when the construction of tall buildings started, and no technology for the construction of deep foundations applicable to the local site conditions is available.

Assume, as above, that the clay layer can support a total  $\Delta\sigma_v$  of no more that **50kPa** without undergoing large deformations, and that each story of a building produces a stress of  $10\text{kN/m}^2$ . Provide an estimate of the number of stories that you would recommend for the Paineiras (large) building to avoid large settlements underneath the centerline of the Mahembi (small) building.

You should not perform additional analyses with Midas. Base your answer on your knowledge of the  $\Delta\sigma_v$  associated with the small building (this value does not change), and take advantage of the fact that the solution provided by Midas is based on elasticity, i.e. stress increments are proportional to the loads applied at the surface. Show your work.

.....

.....